COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Manual de treinamento

V7R2

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Copyright
Copyright (c) Julho 2013 IGE-XAO. Todos os direitos reservados. Nenhuma parte deste manual ou trecho dele pode ser reproduzido, transcrito, salvo ou traduzido, de qualquer forma ou por qualquer meio sem autorização por escrito da IGE-XAO, 25 bld Victor Hugo, Immeuble Le Pythagore, BP 90312 31 773 COLOMIERS CEDEX FRANÇA.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Índice

A INTRODUÇÃO 11

A.1. PROJETO 12
A.2. ARQUIVOS E PASTAS 12

B CRIANDO UM NOVO PROJETO 12

C DESENHANDO A PRIMEIRA PÁGINA DE ESQUEMA ELÉTRICO 12

C.1. CRIANDO A PÁGINA 1 12
C.2. GRID 12
C.3. PÁGINA 1 12
C.4. TRABALHANDO COM O ZOOM 12

D DESENHANDO A SEGUNDA PÁGINA DE ESQUEMA ELÉTRICO 12

D.1. CRIANDO A PÁGINA 2 12
D.2. PÁGINA 2 12
D.3. COPIANDO ELEMENTOS EXISTENTES NA PÁGINA 2 12

E DESENHANDO CABOS 12

F INFORMAÇÕES ADICIONAIS NO ESQUEMA ELÉTRICO 12

F.1. USANDO SÍMBOLOS DE REFERÊNCIA CRUZADA 12
F.2. USANDO SÍMBOLOS DE TEXTO DE INFORMAÇÃO 12
F.3. ÍNDICE DA PÁGINA 12
F.4. TEXTOS 12

G HIPERLIGAÇÕES 12

G.1. ADMINISTRANDO HYPERLIGAÇÕES 12

H LEMBRETES 12

I IMPRESSÃO 12

I.1. IMPRIMIR 12

J LISTAS DE BANCO DE DADOS 12

K LISTAS GRÁFICAS 12

K.1. GERANDO UMA LISTA GRÁFICA 12
K.2. DEFINIR TEXTOS DA PÁGINA PERMANENTEMENTE 12
K.3. GERANDO TODAS AS LISTAS GRÁFICAS EM UM ÚNICO PASSO 12

L INSERINDO OUTROS DOCUMENTOS 12

M CRIANDO SÍMBOLOS 12

M.1. CRIANDO UM SÍMBOLO 12
M.2. MODIFICANDO SÍMBOLOS EXISTENTES 12
M.2.1. APAGANDO ELEMENTOS 12
M.2.2. ADICIONANDO ELEMENTOS 12
M.2.3. ADICIONANDO TEXTOS 12
M.2.4. DESLOCANDO TEXTOS 12
M.3. HANDLING GRAPHICS **ERRO! INDICADOR NÃO DEFINIDO.**
M.4. USING AVAILABLE COMPONENTS **ERRO! INDICADOR NÃO DEFINIDO.**

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

M.5. INFORMATION ABOUT SPECIAL COMPONENTS AND THEIR CREATION **ERRO! INDICADOR NÃO DEFINIDO.**
M.5.1. Contacts *12*
M.5.2. Terminals ***Erro! Indicador não definido.***
M.5.3. Multi-Layer Terminals (Standard) *12*
M.5.4. Connectors ***Erro! Indicador não definido.***
M.5.5. Cross-Reference Symbols ***Erro! Indicador não definido.***
M.5.6. Info Text Symbols ***Erro! Indicador não definido.***
M.5.7. Components with Auxiliary Contacts ***Erro! Indicador não definido.***
M.5.7.a. Symbols with cross reference as text 12
M.5.7.b. Symbols with graphical contact symbols in it **Erro! Indicador não definido.**
M.5.7.c. Examples of Components with Auxiliary Contacts **Erro! Indicador não definido.**
M.5.8. PLC Components *12*
M.5.8.a. Examples of PLC Components 12
M.5.9. Smart black box symbols *12*
M.6. CABLE SYMBOLS 12
M.6.1. Creating User-Defined Cable Symbols *12*
M.6.2. Creating User-Defined Cables *12*
M.6.3. Settings for Cables *12*
M.7. INFORMATION ABOUT COMPONENT INSERTION 12
M.7.1.a. Relay Coils 12
M.7.1.b. Terminals 12
M.7.1.c. Assigning a Component Name 12
M.8. INFORMATION ABOUT SYMBOL DATABASES 12
M.8.1. Creating a New Database *12*
M.8.2. Directories and Names *12*
M.8.3. Working with Symbol Folders *12*
M.8.4. Working with Symbols in the Database *12*
M.8.5. Favorites Symbol Folder *12*
M.8.6. Symbol Database Connection to Modules *12*

N GROUPS 12

N.1. CREATING A GROUP 12
N.2. UNGROUPING SELECTED ELEMENTS 12
N.3. MOTOR TERMINALS AND SIMILAR COMPONENTS 12
N.4. CONNECTORS: USING SEVERAL PLUGS AND JACKS TOGETHER 12

O BASIC OPERATIONS WITH TEMPLATES AND STANDARD SHEETS 12

O.1. CREATING A STANDARD SHEET 12
O.2. CREATING A PAGE TEMPLATE 12
O.3. CREATING A WORKSPACE TEMPLATE 12
O.4. QUICKREFERENCE TEMPLATES AND STANDARD SHEETS 12

P TEMPLATES AND STANDARD SHEETS FOR GRAPHICAL LISTS 12

P.1. CREATING TEMPLATES FOR GRAPHICAL LISTS 12
P.2. FORMATTING TEMPLATES FOR GRAPHICAL LISTS 12
P.3. GENERATE COVER SHEETS FOR GRAPHICAL LISTS 12

Q EXTERNAL DATA TRANSFER 12

Q.1. DATA TRANSFER THROUGH DWG/DXF/DXB FORMAT 12
Q.2. PIXEL IMAGES TRANSFER 12

R PROCESSING A WORKSPACE 12

R.1. ADDITIONAL OPTIONS FOR CROSS-REFERENCES 12
R.2. PLC FUNCTIONALITIES IN THE STANDARD LEVEL 12

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

R.2.1. Using the PLC database *12*
R.3. ADDING AND DELETING PAGES 12
R.4. COPY SINGLE PAGE IN THE SAME WORKSPACE 12

S EASY EDITING IN DATABASE LISTS 12

T CONTACT MIRROR AND TYPE DATABASE 12

T.1. MANIPULATING THE TYPE DATABASE 12
T.2. USING TYPES IN THE CIRCUIT DIAGRAMS 12
T.3. AUXILIARY CONTACTS INSIDE ADD-ONS FOR COILS AND SUBTYPES 12

U ADDITIONAL HINTS ABOUT COMPONENT MANAGEMENT IN THE TYPE DATABASE 12

U.1. CABLE MANAGEMENT 12
U.2. MANAGEMENT OF CONNECTORS 12
U.3. CHANNEL DEFINITION FOR PLC I/OS 12
U.4. BLACK BOXES 12
U.4.1. Assigning of types to intelligent black box symbols *12*
U.4.2. Creating a Black Box from Channel Information *12*

V ADDITIONAL HINTS ABOUT THE TYPE DATABASE 12

V.1. SEARCHING AND COPYING IN THE TYPE DATABASE 12
V.2. COMPLETING COMPONENTS 12
V.3. USING THE COMPONENT EXPLORER 12
V.4. IMPORTING AND EXPORTING ARTICLE MASTER DATA 12
V.5. MODIFYING EXISTING TYPES IN THE PROJECT 12
V.6. DISPLAYING INFORMATION ON THE EQUIPMENT FROM THE TYPE DATABASE 12
V.7. DEFINE WHICH TYPE DATABASE TO USE WITH A PROJECT 12

W WIRE PROPERTIES 12

W.1. WIRE DIRECTION 12
W.2. CHANGING TARGETS 12
W.3. DEFINING A LINK 12
W.4. GRAPHICAL WIRE NUMBERING 12
W.5. SIGNAL PROPERTIES 12
W.6. WORKING WITHOUT SIGNAL PROPERTIES 12
W.7. VIEW AND CHANGE WIRE ATTRIBUTES 12
W.8. DEFINE THE POSITION OF THE WIRE TEXT 12

X TERMINAL MATRIX 12

X.1. GENERATING A TERMINAL MATRIX 12
X.2. HANDLING MULTI-LAYER TERMINALS 12
X.3. CREATING A TEMPLATE FOR A TERMINAL MATRIX 12
X.3.1. General Information *12*
X.3.2. Drawing Graphics Associated with Each Terminal – One Geometry for All Terminals *12*
X.3.3. Drawing Graphics Associated with Each Terminal – One Specific Geometry for Each Kind of Terminal 12
X.4. DRAW MORE THAN ONE TERMINAL STRIP PER PAGE 12

Y CABLE PLAN WITH GRAPHICS 12

Y.1. CREATING A CABLE PLAN WITH GRAPHICS 12
Y.2. CREATING A TEMPLATE FOR CABLE PLANS WITH GRAPHICS 12
Y.3. SHOWING NOT USED CABLE CORES IN CABLE PLAN 12
Y.4. CABLE PLAN WITH SHIELD INFORMATION FOR CABLES 12

Z WIRING LISTS 12

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Z.1. GRAPHICAL WIRING LIST 12
Z.1.1. Creating an Excel/Text File 12

AA FUNCTION AND LOCATION 12

BB REVISION MANAGEMENT 12

CC ADVANCED FUNCTION/LOCATION AND PRODUCT MANAGEMENT 12

CC.1. DATABASE FOR FUNCTION/LOCATION AND PRODUCT MANAGEMENT 12
CC.2. USING NESTED ASPECTS 12

DD COMPLEX MODIFICATIONS OF THE DATABASE LISTS 12

DD.1.1. Product Editor 12
DD.1.2. Terminal Editor 12
DD.1.3. Cable Editor 12
DD.1.4. Connector Editor 12
DD.1.5. Signal Editor 12
DD.1.6. Wire Editor 12
DD.1.7. Function/Location Editor 12
DD.1.8. Document Editor 12
DD.1.9. REDLINING OBJECTS LIST 12

EE ADVANCED PROCESSING OF A PROJECT 12

EE.1. NAVIGATION IN THE PROJECT 12
EE.2. WORKING WITH AUTOMATICALLY GENERATED CONNECTIONS 12
EE.3. ORTHOGONAL WIRING 12
EE.4. ADVANCED FUNCTIONALITY FOR CABLES 12
EE.4.1. Define Cable Names Automatically 12
EE.4.2. Display Cable Cores a Second Time 12
EE.5. ADVANCED FUNCTIONALITY FOR POTENTIAL NUMBERING 12
EE.6. COPYING PAGES 12
EE.6.1. Copy Single Page in the Same Workspace 12
EE.6.2. Copy Single Page in the Same Workspace or in Different Workspace 12
EE.6.3. Copy Multiple Pages Between Different Workspaces 12
EE.7. CHANGING PAGE TEMPLATES 12
EE.8. TRANSLATING A WORKSPACE 12
EE.8.1. Displaying Several Translations at the Same Time 12
EE.8.2. Translation of Part Strings 12
EE.8.3. Updating Translation Text in a Sheet 12
EE.8.4. Looking up Texts in the Translation Database 12
EE.8.5. Exchanging the Source Languages of a Workspace 12
EE.9. CHANGING THE FONTS AND ATTRIBUTES FOR ALL TEXTS IN THE WORKSPACE, SYMBOL LIBRARIES OR PAGE TEMPLATES 12
EE.10. PLC FUNCTIONALITIES IN THE ADVANCED LEVEL 12
EE.11. GENERATE GRAPHICAL LISTS IN ONE STEP 12

FF CUSTOMIZING THE WORKSPACE/PAGE INFORMATION WINDOWS 12

GG CUSTOMIZING THE WORKSPACE TREE 12

GG.1. HIDE UNNECESSARY DATABASE AND GRAPHICAL LISTS 12
GG.2. SORTING OF DOCUMENTS IN THE WORKSPACE TREE 12

HH USER-DEFINED SQL-QUERIES 12

HH.1.1. Creating an SQL-Query for Adding a Database List in the Workspace Explorer 12
HH.1.1. Definition of Graphical Formulas 12

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

II TERMINAL PLAN 12

II.1. GENERATING A TERMINAL PLAN 12
II.2. CREATING A TEMPLATE FOR A TERMINAL PLAN 12
II.2.1. Drawing Graphics Associated with Each Terminal 12
II.2.2. DESENHANDO CABOS as a Grouping of Cable Cores 12
II.2.3. DESENHANDO CABOS/Geometry on Both Sides of the Terminals 12
II.2.4. Showing that Cable Cores are Used on Previous/Next Page 12
II.2.5. Symbols for a Terminal Plan 12

JJ CONNECTOR MATRIX AND PLAN 12

JJ.1. GENERATING A CONNECTOR MATRIX 12
JJ.2. CREATING A TEMPLATE FOR A CONNECTOR MATRIX 12
JJ.3. DRAW MORE THAN ONE CONNECTOR PER PAGE 12
JJ.4. CONNECTOR PLAN 12

KK TERMINAL ROW PICTURE PLAN 12

KK.1. GENERATING A TERMINAL ROW PICTURE PLAN 12

LL CABLE TERMINAL ROW PLAN 12

MM PRODUCT ASSEMBLY LIST 12

MM.1. GENERATING A TEMPLATE FOR THE PRODUCT ASSEMBLY LIST 12

NN MULTICORES 12

NN.1. USING MULTICORES 12

OO INSERTING COMPONENTS NOT IN THE DRAWING VIA THE DATABASE EDITORS 12

OO.1. EDITOR FOR COMPONENTS WITHOUT GRAPHICS 12
OO.2. INSERTING SPARE OR GROUND TERMINALS VIA THE TERMINAL EDITOR 12
OO.2.1. Handling Cables Without Graphics 12
OO.3. INSERTING SYMBOLS FOR COMPONENTS WITHOUT GRAPHICS IN THE DIAGRAM (PICK LIST) 12

PP AUTO DIAGRAM 12

PP.1. INTRODUCTION 12
PP.2. CREATING SYMBOLS (GROUPS) 12
PP.3. PAGE TEMPLATES 12
PP.4. EXCEL SPREADSHEET 12
PP.4.1. Project Data (ProjectInfo) 12
PP.4.2. Page Data (PageInfo) 12
PP.4.3. Select Symbol (Symbols) 12
PP.4.4. Defining Alias Names 12
PP.5. AUTOMATIC GENERATION OF CIRCUIT DIAGRAMS 12
PP.5.1. The Autodiagram Command 12

QQ LISTS AND LABELS EDITOR 12

QQ.1. CREATING LABEL TEMPLATES 12
QQ.1.1. Files for Label Templates 12
QQ.1.2. Changing Label Templates 12
QQ.1.3. Printing Labels 12
QQ.2. CREATING LIST TEMPLATES 12
QQ.2.1. Files for List Templates 12
QQ.2.2. List Title (Text Object) 12
QQ.2.3. Header Lines 12
QQ.2.4. Group Header 12

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

QQ.2.5. Group Footer Lines 12
QQ.2.6. Data Lines 12
QQ.2.7. Footer Line 12
QQ.2.8. List Footer Line (Text Object) 12
QQ.3. EDITING LISTS 12
QQ.3.1. Changing Column Titles (Header Lines) 12
QQ.3.2. Changing a List (Font Size, Fields Order, Adding or Deleting Fields) 12
QQ.3.3. Changing the List Title 12
QQ.3.4. List Title with Questions 12
QQ.3.5. List Title with Generation Date 12
QQ.3.6. Filtering in the Lists 12
QQ.3.6.a. List of Parts for a Particular Manufacturer 12
QQ.3.6.b. List of Parts without Terminals 12
QQ.3.7. Calculations 12
QQ.3.7.a. Calculation of Order Costs 12
QQ.3.8. Line Numbering 12
QQ.4. DEFINING SQL-QUERIES FOR LISTS AND LABELS 12
QQ.4.1. Joining Two Lists 12
QQ.4.1.a. Joining Information from Cable List and Cable-Wires List 12
QQ.4.2. Filtering Double Records 12
QQ.4.2.a. List of Functions/Locations on Pages 12
QQ.4.3. Sorting a List 12
QQ.4.3.a. Order List 12
QQ.4.3.b. Cable Length Addition 12
QQ.4.3.c. Length Calculation for Cable Channels and Rails 12
QQ.4.4. Multiple Printing of Labels 12
QQ.4.5. Labels for Component Names with Defined Content 12
QQ.5. USEFUL TABLES IN ACCESS DATABASE 12
QQ.6. STATEMENT SUMMARY AND COMMANDS IN LISTS AND LABELS 12
QQ.6.1. SQL Statements 12
QQ.6.1.a. SELECT 12
QQ.6.1.b. SELECT DISTINCT 12
QQ.6.1.c. JOIN –clause 12
QQ.6.1.d. ORDER BY 12
QQ.6.1.e. WHERE/WHERE NOT 12
QQ.6.1.f. UNION ALL 12
QQ.6.1.g. Uppercase/ Lowercase 12
QQ.6.1.h. Logical Operators 12
QQ.6.1.i. Relational Operators 12
QQ.6.2. Commands in Lists and Labels 12
QQ.6.2.a. Conditions: If 12
QQ.6.2.b. Upper$/Lower$ 12
QQ.6.2.c. Val 12
QQ.6.2.d. Fstr$ 12
QQ.6.2.e. AskString$ 12
QQ.6.2.f. Len, Left$, Right$ 12
QQ.6.2.g. Relational Operators 12

RR CABINET LAYOUT 12

RR.1. DRAWING CABINET LAYOUTS 12
RR.1.1. With the Cabinets Module 12
RR.1.2. Without the Cabinets Module 12
RR.2. USING LAYERS 12

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

RR.3. DIMENSION 12
RR.4. COMPARISON BETWEEN CIRCUIT DIAGRAMS AND CABINETS 12

SS USEFUL TOOLS 12

SS.1. COMPRESSING PROJECTS 12
SS.2. REORGANIZE WORKSPACES 12

TT SETTINGS 12

TT.1. SYSTEM SETTINGS 12
TT.2. WORKSPACE PROPERTIES 12
TT.3. PROPERTIES FOR CIRCUIT DIAGRAMS 12
TT.4. PAGE PROPERTIES 12
TT.5. CUSTOMIZING THE INTERFACE 12
TT.5.1. Quick Access Toolbar 12
TT.5.2. Define Hotkeys 12
TT.5.3. User Defined Categories 12

UU COMMANDS EXPLORER 12

VV INTELLIGENT DRAWING LEGACY MODULE 12

VV.1. TOOLS FOR SCANNED DRAWINGS 12
VV.1.1. Importing Multiple Scanned Drawings in One Step 12
VV.1.2. Covering Scanned Symbols with an (White) Area ("BkSymbol" Command) 12
VV.2. TOOLS FOR MAKING DXF/DWG IMPORTED DRAWINGS MORE INTELLIGENT 12
VV.2.1. Defining Patterns ("DefinePatterns" Command) 12
VV.2.2. Controling and Changing Patterns, Assigning Texts ("ShowPatterns" Command) 12
VV.2.3. Recognizing Patterns ("RecognizeDrawingPatterns" and "RecognizeWorkspacePatterns" Commands) 12
VV.2.4. Displaying all Elements with Logic 12

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

A INTRODUÇÃO

Ao seguir este manual passo a passo você ganhará experiência no uso do *SEE Electrical*.
Caso não tenha trabalhado com o *SEE Electrical*, você poderá seguir os exemplos dos capítulos subsquentes. Com isso sera possível aprender as funções fundamentais do *SEE Electrical*.

Os primeiros capítulos possuem informações sobre funções que serão usadas mais adiante neste *Manual de Treinamento*.
Os capítulos "*Edição simples das listas de banco de dados*" a "*Função e localização*" aplicam-se ao *SEE Electrical Standard*, de "*Modificações complexas das listas de banco de dados* " a "*Editor de lista e etiqueta*" aplicam-se apenas ao *SEE Electrical Advanced*. O capítulo "*Layout*" necessita do *SEE Electrical Standard,* e alguns exemplos requerem o respective módulo.

Abreviações usadas neste manual de treinamento:

CA	Seleção de uma categoria
CO	Seleção de um comando
M	Seleção a partir de um menu suspenso
+	Seleção de um elemento com o cursor
#	Entrada pelo teclado
>	Seleção de campo em uma janela
<Input>	Digitação de texto ou seleção de um elemento.
T	Seleção de aba
I	Seleção de ícone na barra de ferramentas

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

A.1. PROJETO

Um projeto do *SEE Electrical* nornalmente contém diagramas elétricos. As listas gráficas são geradas automaticamente usando as informações dos diagramas. Seguem alguns exemplos:

- ✓ Lista de componentes
- ✓ Lista de bornes
- ✓ Lista de CLP
- ✓ Lista de fiação
- ✓ Lista de cabos
- ✓ Lista de documentos
- ✓ etc.

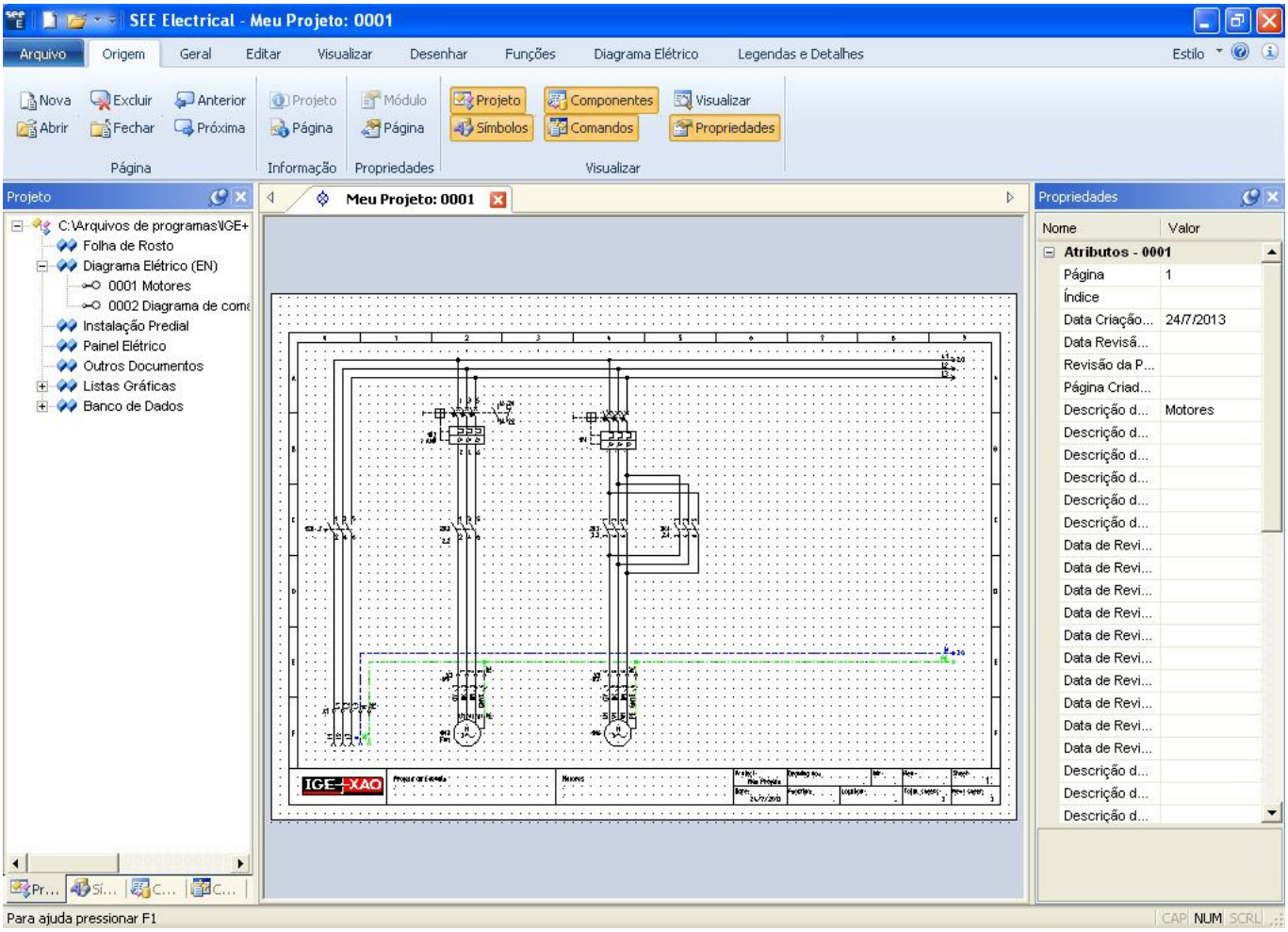
Os dados do projeto são usados para geração da lista gráfica de bornes (assim como a régua de bornes no nível *Standard*), de cabos e componentes.
Você também pode criar desenhos de paineis.

O *SEE Electrical* contém diferentes módulos que disponibilizam funções para o desenho de diagramas elétricos, instalações prediais ou layout. A disponibilidade do modulo apropriado permitirá a você a execução dos exemplos a seguir.

Outros documentos podem ser adicionados nos projetos a partir de ***Outros documentos***, e podem incluir, por exemplo, documentos do *Word* ou planilhas do *Excel*.
Por padrão, três abas aparecerão ao iniciar o *SEE Electrical*.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

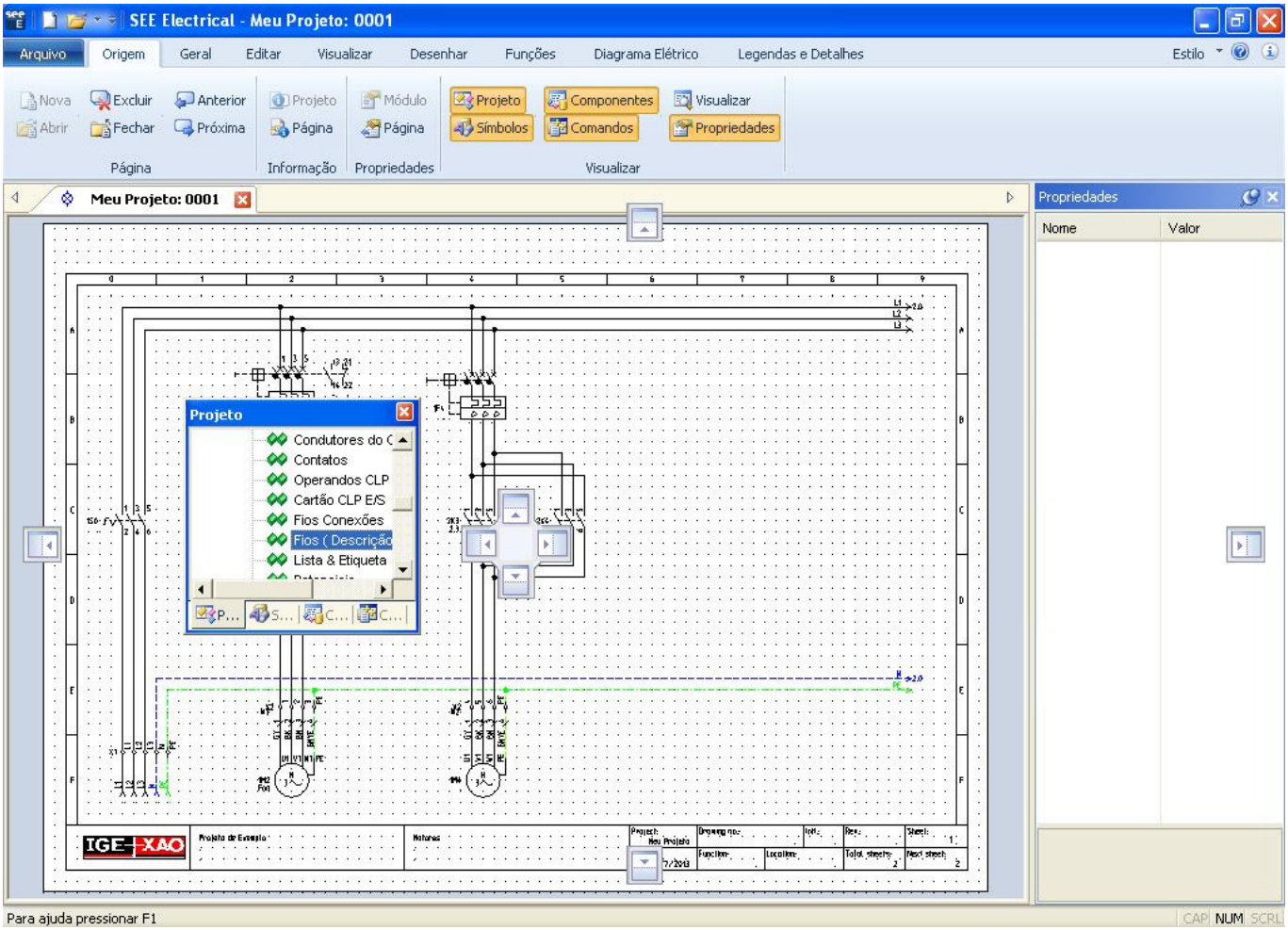
As abas Projeto/Símbolos/Comandos estão localizadas no lado esquerdo da janela do *SEE Electrical*. A area de desenho está localizada no centro da janela do *SEE Electrical*. A aba ***Propriedades*** está localizada no lado direito da janela do *SEE Electrical*.



Para visualizar uma lista, vá para a guia ***Projeto***, expanda o item Banco de dados e selecione a lista desejada com um duplo clique.
Você pode alternar entre as abas ***Projeto***, ***Símbolos***, ***Componentes*** e ***Comandos***.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Você pode modificar o posicionamento das abas ***Projeto***/***Símbolos***/***Propriedades*** e ***Comandos*** arrastando-as e soltando-as no local desejado. Use as setas que aparecerão na tela para soltá-las na sua nova posição.



Para visualizar uma determinada area da página aberta, clique em **Origem** e ative o comando ***Visualizar***. A janela ***Visualizar*** aparecerá e você poderá ampliar ou diminuir o zoom na página. Veja o item ***Trabalhando com o zoom*** para mais detalhes.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

A.2. ARQUIVOS E PASTAS

O *SEE Electrical* utiliza as seguintes pastas e arquivos:

<pasta SEE Electrical V7 >	Contém os arquivos do programa *SEE Electrical*.
...\PROJECTS	Nesta pasta são armazenados por padrão os projetos do *SEE Electrical*. Os arquivos de projetos possuem a extensão *.SEP*.
...\SYMBOLS	Nesta pasta são armazenados os bancos de dados de símbolos do *SEE Electrical*. As bibliotecas de símbolos possuem a extensão .SES. O banco de dados TYPES.SES, necessário para os níveis *Standard* e *Advanced* também é armazenado nesta pasta. Tenha em mente que a biblioteca *SYSTEM.SES* é necessária pelo o programa internamente e não deve ser removida desta pasta. Se o modulo de Layout estiver disponível o arquivo IndexTable.SES é usado para criar tabelas de indexação.
....\TEMPLATES	Esta pasta contém modelos para projetos e formulários, e modelos para listas e etiquetas. As fontes também são salvas nesta pasta. *SEP*: Modelos para projetos *TDW*: Páginas modelo *CABLESNEW.MDB ou CABLESNEWIEEE.MDB possuem os padrões para cabos.* *DAT*: Fontes
...\TEMPLATES\LABEL_SETTINGS	*Arquivos SLS* (usados para criar etiquetas no formato de diversas impressoras).

Além disso, o *SEE Electrical Advanced* utiliza os seguintes arquivos:

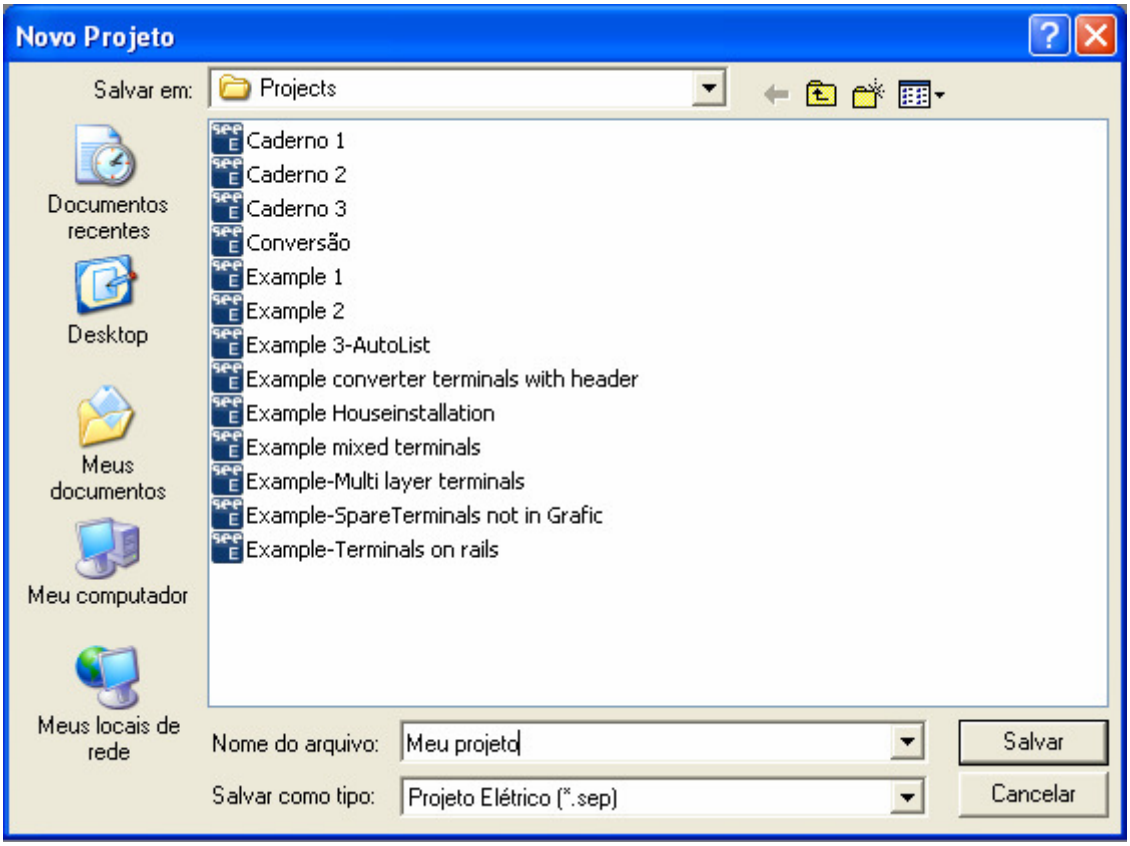
...\TEMPLATES	Nesta pasta está o arquivo de banco de dados de tradução *TRANSLATIONNEW.MDB* usado pelo *SEE Electrical Advanced* para a tradução de projetos.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

B CRIANDO UM NOVO PROJETO

Exercício 2-1: Criar um novo projeto.

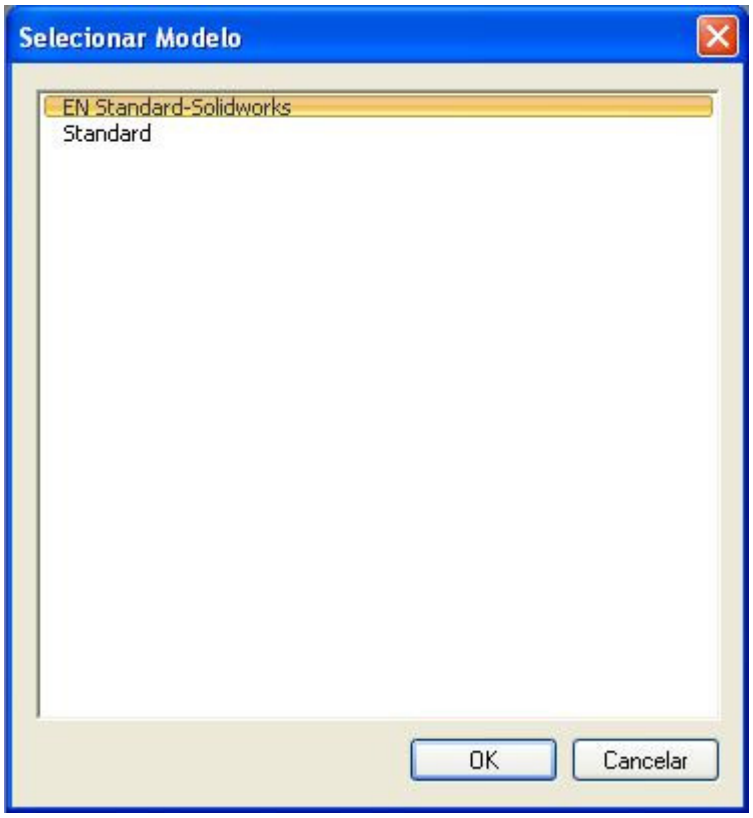
1.CA **Arquivo**
2.CO **Novo**



3.> Nome do arquivo
4.# Meu Projeto
Você poderá dar um novo nome ao projeto, se desejar.
5.> **Salvar**

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

O projeto será criado. Uma lista de modelos para projetos aparecerá.



6.> <Selecionar modelo>
Selecione um modelo para projetos.
Um modelo para projetos contém, entre outras informações, os formulários que por sua vez possuem também a configuração do número de colunas no desenho, etc.
A instalação do *SEE Electrical* inclui ao menos um modelo para projetos. Selecione o modelo **Standard**.

7.> Clique em **OK**.
O projeto novo será aberto. A aba ***Propriedades***, visível no lado direito da tela contém as informações sobre o projeto.
Em "*Nome do arquivo*" você poderá ver o nome e localização do arquivo do projeto (<nome>.SEP).

8.> Descrição do projeto linha 01

9.# Projeto de exemplo

10.> Data de criação do projeto
Se o campo ***"Data de criação do projeto"*** não estiver visível, utilize a barra de rolagem.
O campo ***"Data de criação do projeto"*** é preenchido automaticamente.

11.# Digite outra data, caso necessário.
O *SEE Electrical* mostrará a data no campo **Data de criação do projeto.**
Se você quiser usar um formato de data diferente, deverá modificar as configurações do seu sistema operacional. Quando o formato de data do computador for diferente do usado no projeto a conversão não sera possível e o software mudará seu formato para texto simples de forma a garantir que a informação original possa ser visualizada.

12.> Projeto criado por

13.# Digite seu nome.
Você poderá preencher os demais campos conforme quiser.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

O conteúdo dos campos da aba propriedades do projeto serão inseridos automaticamente no formulário de diagramas elétricos, caso os campos existam nele.

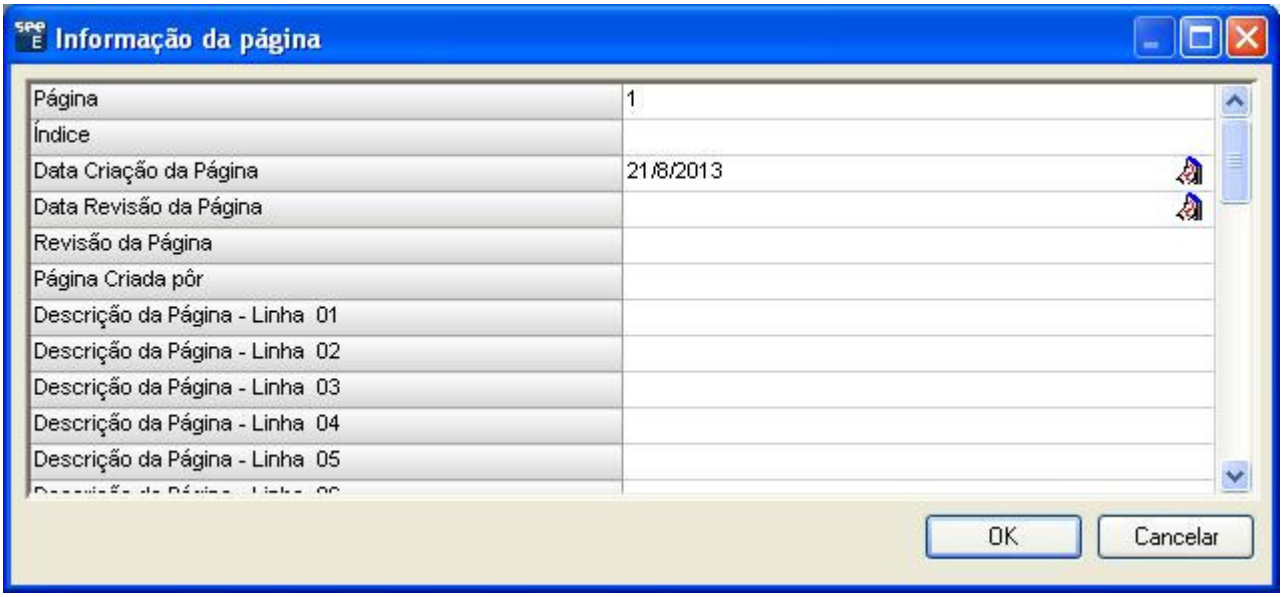
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

C DESENHANDO A PRIMEIRA PÁGINA DE ESQUEMA ELÉTRICO

C.1. CRIANDO A PÁGINA 1

Exercício 3-1: Criação da primeira página do projeto.

1.> Criar página
Na parte inferior da aba Propriedades, selecione “Diagrama Elétrico” e clique no botão Nova Pá... .
A caixa de diálogo ***Informação da página*** aparecerá, permitindo a inserção de novos dados.



2.> Informação da página
Página: "1" (inserido automaticamente).
Por padrão, o *SEE Electrical* oferecerá o primeiro número disponível no respectivo módulo, neste caso, 1. Você poderá modificar essa informação, se necessário.

3.> Data de criação da página
O *SEE Electrical* insere automaticamente a data. Você poderá modificá-la clicando no botão presente neste campo.
Há outras informações que podem ser preenchidas caso necessário.

4.> Descrição da página linha 01

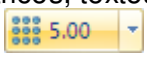
5.# Diagrama de força

6.> **OK**
A caixa de diálogo será fechada.
O *SEE Electrical* abrirá uma nova página de diagramas elétricos.
As informações preenchidas anteriormente serão inseridas automaticamente na página, caso os campos existam no formulário.
Nos passos a seguir você começará a desenhar o esquema elétrico.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

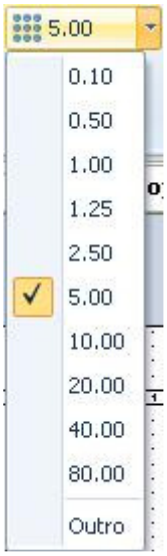
C.2. GRID

Usando o grid você poderá alinhar precisamente elementos geométricos, textos e componentes.

É possível ligar ou desligar a visualização do grid clicando no ícone 

da aba **Desenhar**.

Após clicar no botão , uma lista de valores de grid aparecerá:



Escolhendo a opção "*Outro*", é possível definir manualmente o valor do grid.

Dica:

Também é possível fazer com que valores personalizados apareçam no menu de de grid. Para fazer isso é necessário modificar o arquivo ***CAEGridSettings.xml****, que está na pasta raiz de instalação do programa. Este arquivo não pode ter mais de dez valores de grid.*

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

C.3. PÁGINA 1

Exercício 3-2: Inserir a entrada de alimentação.

- Clique na guia ***Símbolos***, presente no lado esquerdo da tela do *SEE Electrical*.

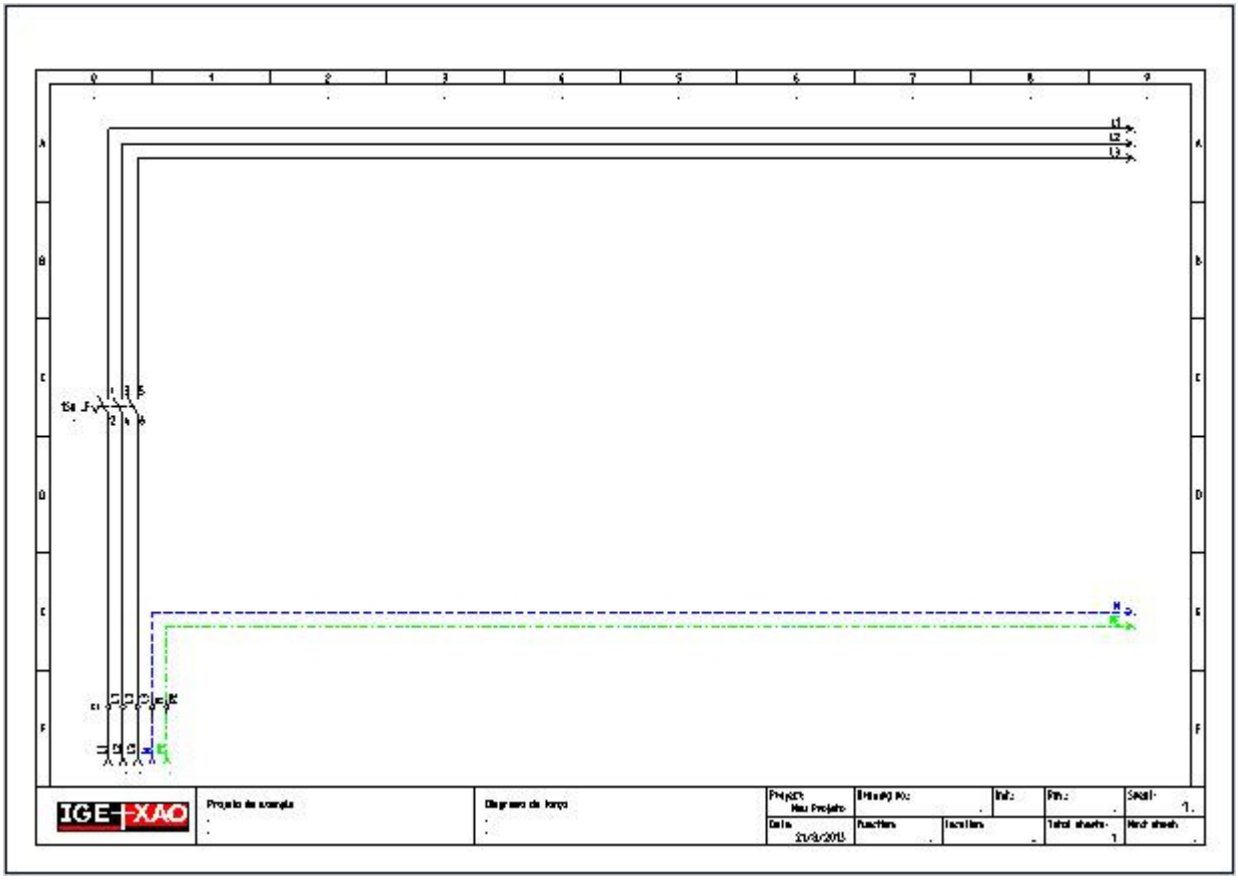
Esta parte da tela contém as guias ***Projeto***, ***Símbolos***, ***Componentes*** e ***Comandos***.
Caso ela não esteja visível é possível ativá-la a partir da barra de ferramentas **Visualizar** da categoria **Origem**.

Exercício 3-3: Selecionar o banco de dados que contém os símbolos a serem utilizados. Neste caso, **Examples**.

1.T Ative a guia ***Símbolos***.
2. Dê um duplo clique no banco de dados de símbolos *Examples*.
   O *SEE Electrical* abrirá o banco de dados de símbolos.
   Várias pastas de símbolos serão mostradas.
3. Dê um duplo clique na pasta **Examples** para abrí-la.
   Todos os símbolos serão mostrados.
4. Clique no símbolo **Power supply**.

   O símbolo aparecerá ligado ao cursor.

5. "Arraste" o símbolo para a posição tendo como exemplo a figura abaixo.



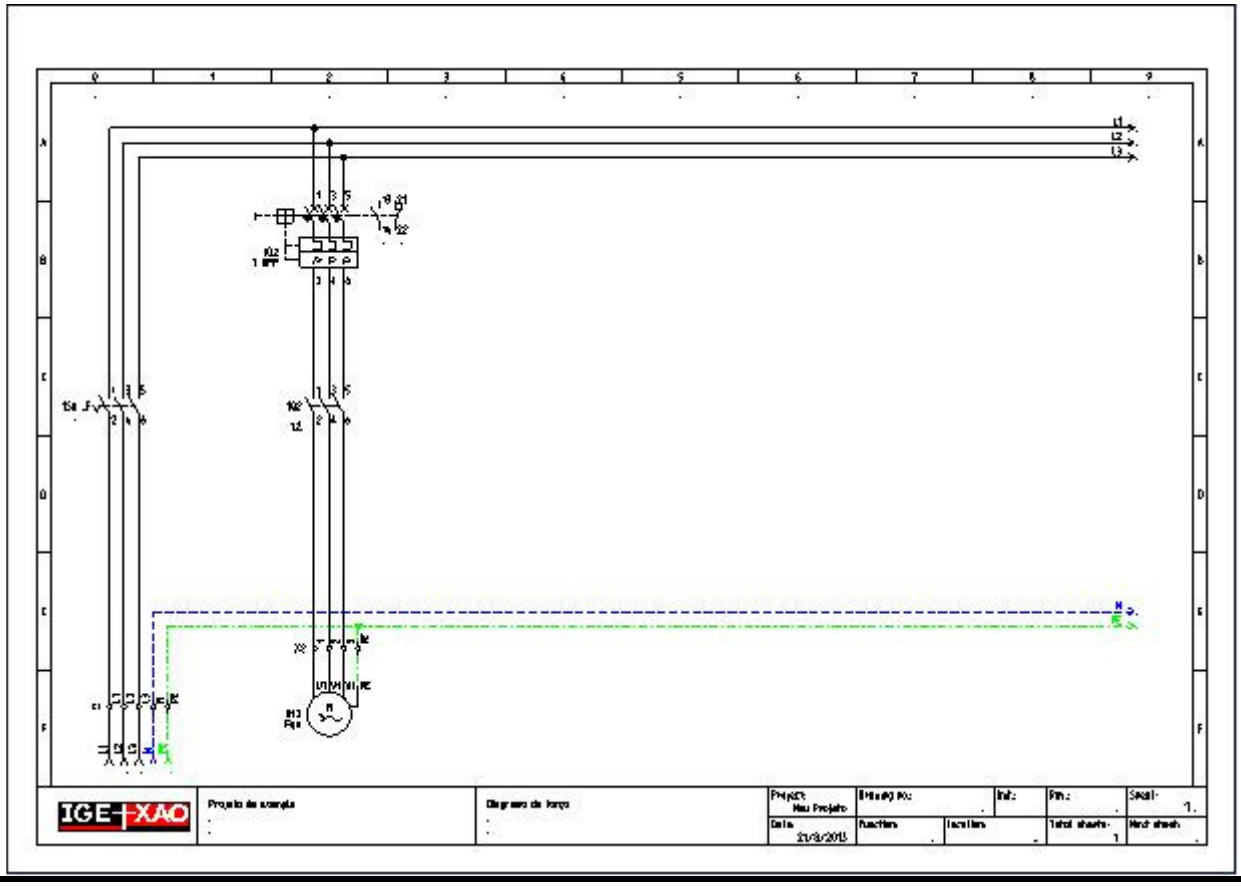
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

6.+ "Solte" a entrada de alimentação na coluna 0.
7.> Component (-)
8.# X1
Insira o nome da régua de bornes.
Não modifique o número do borne.
9.> **OK**
Os próximos bornes terão a régua definida automaticamente como X1.
Uma caixa de diálogo com o nome dos potenciais aparecerá.
10.> Componente (-)
11.# N
O nome do potencial sera sugerido.
12.> **OK**
Clique em **OK** para aceitar o nome sugerido.
Use o mesmo procedimento e aceite os demais nomes sugeridos para os próximos quarto potenciais.
Clique com o botão direito do mouse para sair do modo de inserção.

Exercício 3-4: Salve o projeto.

1.CA **Arquivo**
2.CO **Salvar**

Exercício 3-5: Inserir o diagrama de força.

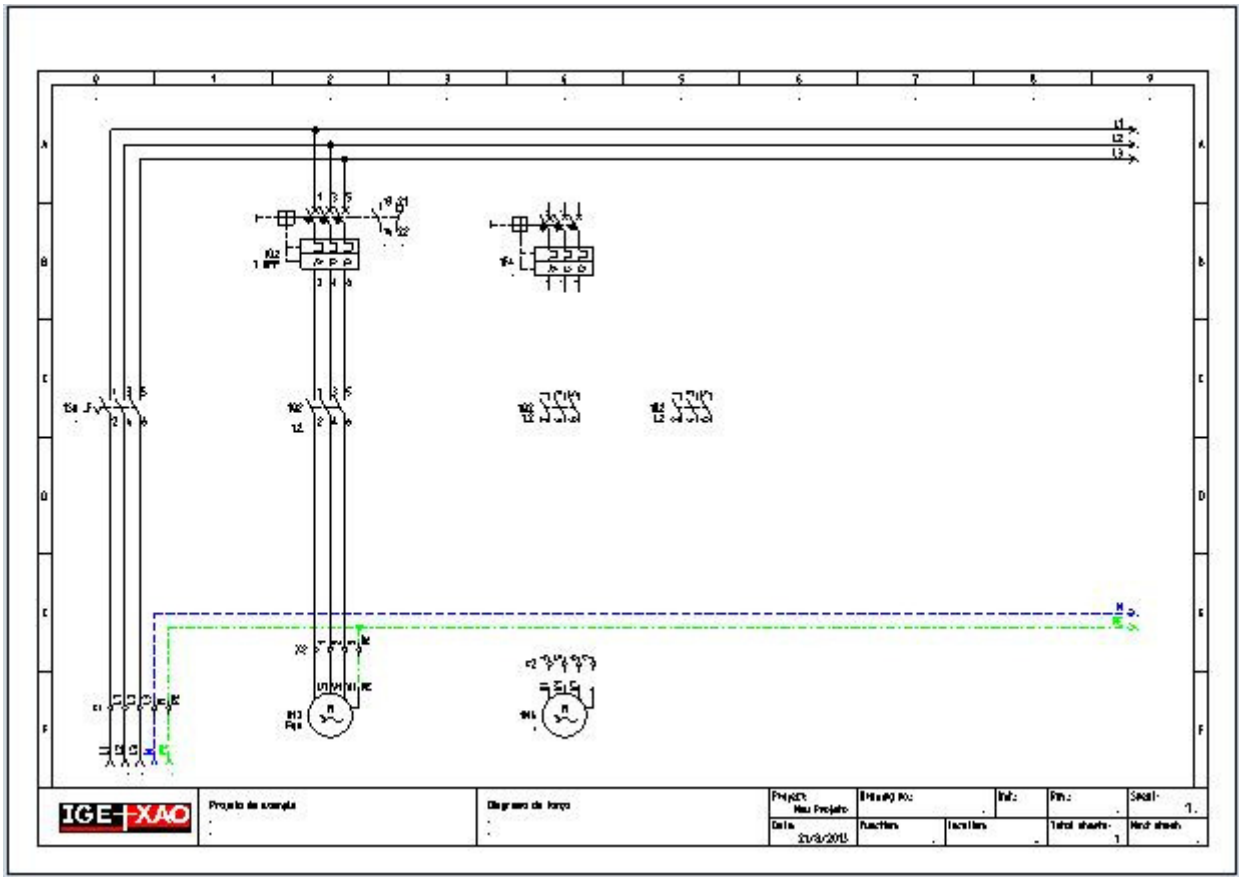


COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

1. Movimente o cursor para a guia ***Símbolos***.
2. Clique no grupo **Three-phase motor direct** presente na pasta *Example*.

3. Arraste o grupo **Three-phase motor direct** para a área de desenho conforme a figura acima.
4.+ Solte o grupo na posição desejada.
Uma caixa de diálogo de nome dos bornes aparecerá.
5.> Componente (-)
6. # X2
O nome X1 será sugerido. Utilize X2.
7.> **OK**
Os próximos bornes serão vinculados automaticamente a essa régua.
A caixa de diálogo para os contatos de força NA aparecerá.
8.> Componente (-)
9.# 1Q2
Você pode inserir o nome neste contato, mas não o modifique agora pois ainda não sabe a qual componente pertence.
10.> **OK**
Clique com o botão direito do mouse para sair do modo de inserção.

Exercício 3-6: Inserir os símbolos do Segundo diagrama de força.



1. Movimente o cursor para a guia ***Símbolos***.
2. Abra a biblioteca **EN61346-2UK**.

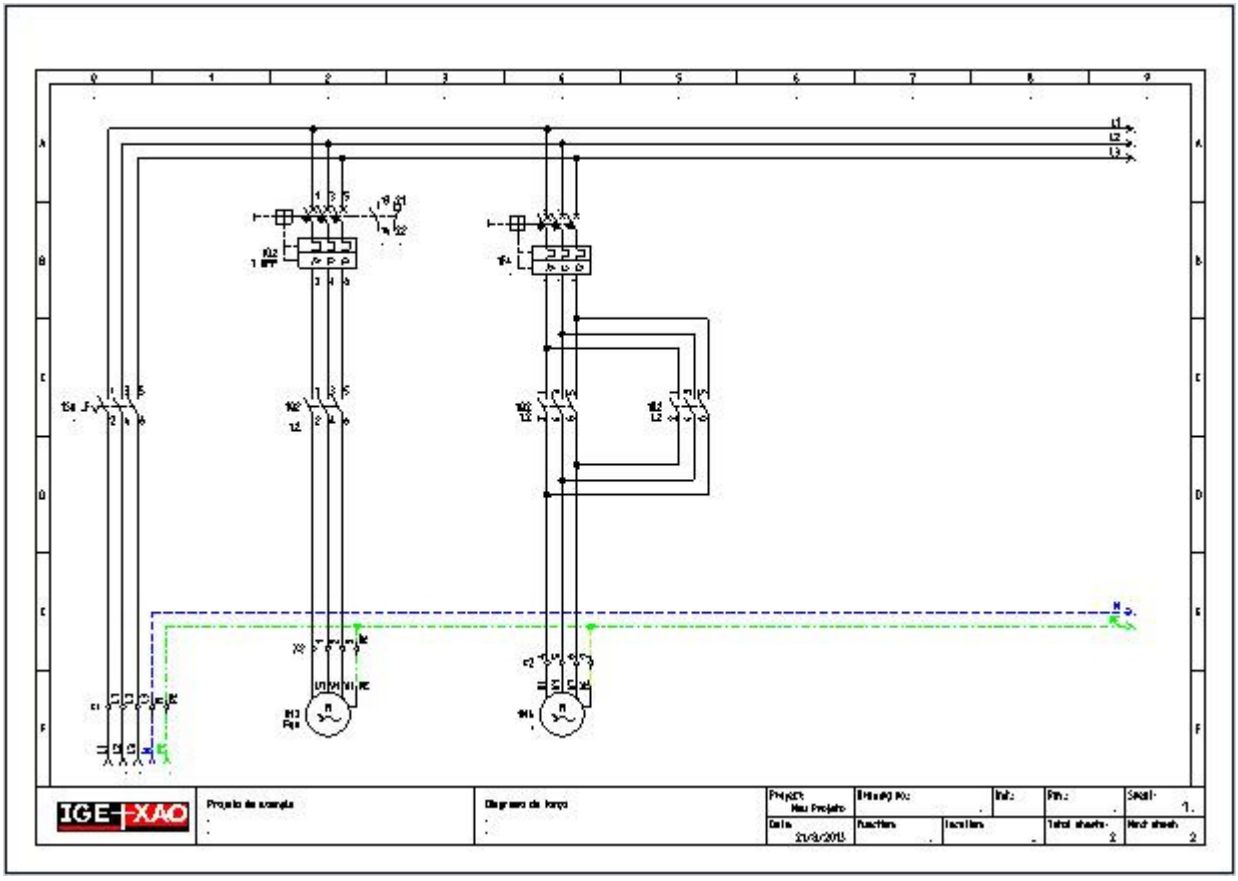
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

3. Abra a pasta *Motors and Generators***.** Selecione o símbolo *Three-phase + PE* e arraste-o para a area de desenho.
4.+ Clique na coluna 4 para soltá-lo.
5. Dê um duplo clique na pasta *Relay-contacts, MAIN* para abrí-la e selecione o símbolo "*3-pole NO*".
6. Arraste o símbolo com o cursor para para a coluna 4. Clique para soltar o símbolo.

7.> A caixa de diálogo ***Propriedades do componente*** aparecerá.
8.> Componente (-)
9.# 1Q2
Você pode inserir o nome neste contato, mas não o modifique agora pois ainda não sabe a qual componente pertence.
10.> **OK**
Feche a caixa de diálogo.
11.> Insira o segundo contato na posição desejada a proceda da mesma maneira.
A caixa de diálogo ***Propriedades do componente*** aparecerá novamente. Mantenha o nome sugerido.
12.> Clique em OK
A caixa de diálogo será fechada.
13. Abra a pasta **Terminals** e selecione o símbolo **4 terminals 90° vertical**. Insira-o na coluna 4) procedendo da mesma maneira sugerida nos passos anteriores.
A caixa de diálogo para definição do nome dos bornes aparecerá.
14.> Componente (-)
15. # X2
16.> Número do borne
17. # 4
18.> Número de ordenação
19. # 5
20.> **OK**
Os próximos bornes serão automaticamente vinculados a essa régua.
21. Abra a pasta *Protective devices*.
Selecione o símbolo "*3-pole trip breaker*" e arraste-o para a guia 4. Solte o símbolo na posição desejada clicando com o mouse.
22. Clique com o botão direito do mouse para sair do modo de inserção.
Todos os símbolos terão sido inseridos.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercício 3-7: Desenhar a conexão trifásica.



1. Ative o comando **Desenho elétrico Conexão dos fios 3 fios**.
2. Clique para definir o ponto inicial da conexão no potencial L1 e de forma alinhada com com as conexões do primeiro borne (X2:4).
3. Clique para definir o segundo ponto na conexão superior do primeiro borne (X2:4).
   As três conexões serão desenhadas automaticamente: entre o potencial L1 e o primeiro borne (X2:4), entre o potencial L2 e o segundo borne (X2:5) e entre o potencial L3 e o terceiro borne (X2:6).
   As conexões são interrompidas automaticamente onde há símbolos.
4. Agora use a função **Desenho elétrico Conexão dos fios Ligação ortogonal** para desenhar uma conexão trifásica entre os bornes e o motor.
   Clique para definir o início do fio na conexão inferior do borne X2:4.
   Clique novamente para definir o final na conexão U1 do motor.
   Quatro fios serão desenhados.
5. Continue a desenhar conexões ortogonais conforme abaixo:
   Clique na conexão entre o disjuntor e os contatos e tomando como referência o potencial L3. Depois disso, clique na **conexão 6** do segundo jogo de contatos NA.
   Igualmente **clique na conexão 1** dos mesmos contatos NA e depois na conexão entre o primeiro jogo de contatos NA e os bornes.
6. Clique com o botão direito do mouse para sair do modo de desenho.

Dicas

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

1. *Ao mover o cursor, uma visualização dinâmica da conexão e pontos de conexão aparecerá, permitindo a você verificar se os fios que está tentando desenhar estão sobrepondo outros já existents ou mesmo se há algum problema com relacão a símbolos já inseridos. Neste caso, o desenho é verificado automaticamente pelo SEE Electrical quanto as conexões possíveis.*

Exercício 3-8: Desenhar o fio faltante ente o potencial PE e o borne X2:7. Modificar o estilo da linha e cor do fio da conexão entre o potencial PE e o motor M2.

1.+ Use a função **Diagrama Elétrico** □ **Conexão dos fios** □ **1 Fio** conforme descrito para criar a conexão faltante entre o potencial PE e o borne X2:7.
2.+ Clique com o botão direito do mouse sobre o fio entre o potencial PE e o borne X2:7 e selecione **Propriedades** a partir do menu suspenso.
As propriedades do fio aparecerão na aba ***Properties*** no lado direito da janela do *SEE Electrical*.
3.> Estilo da caneta
4.> Selecione o estilo "Tracejado".
5.> Cor da caneta
6.> Selecione a cor verde.
A cor e estilo do fio serão modificados dinamicamente.
Repita a mesma operação para o fio entre o borne X2:7 e o motor.
7. Clique no motor 1M4. Suas propriedades aparecerão na respective aba. Preencha "PE" no campo da conexão 03. Tecle "Enter" para validar o operação.
8. Selecione o borne X2:7 e, conforme descrito acima, mude seu número para PE na guia ***Propriedades***.

A primeira página de esquema elétrico está completa.

Exercício 3-9: Salve o projeto.

1.CA **Arquivo**
2.CO **Salvar**

Salve seus projetos frequentemente. Isso também é possível clicando no ícone .

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

C.4. TRABALHANDO COM O ZOOM

Com muita frequência é necessário trabalhar com uma visão mais detalhada do desenho.

Exercício 3-10: Efetuando zoom por meio da janela ***Visualizar***.

Na janela ***Visualizar***:

1.+ Clique o primeiro ponto do retângulo da área que você deseja definir o zoom.
O retângulo é definido por dois pontos diagonalmente opostos.
2.+ Clique o segundo ponto do retângulo.
A visualização na tela principal será expandida.
Você poderá mover a área expandida na janela ***Visualizar***
3+ Posicione o cursor no retângulo cinza que define a área com visualização ampliada.
O cursor será representado por uma mão.
4+ Clique e arraste com o botão esquerdo do mouse o retângulo.
Para desativar o zoom, tecle o comando **Zoom Original** por meio do atalho F3.

Exercício 3-11: Aumentar a visualização de uma parte do esquema elétrico.

1.CA **Visualizar**
2.CO **Zoom Janela**
3.+ Clique o primeiro ponto do retângulo da área que você deseja aumentar a visualização.
O retângulo é definido por dois pontos diagonalmente opostos.
4.+ Clique o segundo ponto do retânguilo.
Você pode ativar o **Zoom Janela** teclando o atalho F4.

Exercício 3-12: Retorne a visualização original do diagrama elétrico.

1.CA **Visualizar**
2.CO **Zoom Original**
Você poderá ver o desenho completo novamente.
É possível ativar o comando **Zoom Original** teclando o atalho F3.

Exercício 3-13: Movendo a visualização ampliada.

1.CA **Visualizar**
2.CO **Zoom Panorâmico**
O cursor aparecerá como uma mão.
3.+ Clique com o botão esquerdo do mouse e a arraste para o local desejado.
É possível ativar o comando **Zoom Original** teclando o atalho F3.

Dica:

É possível definir o zoom com o botão de rolagem do mouse mantendo pressionada a tecla CTRL e movimentar a rolagem para cima (aumentar) ou para baixo (diminuir).
Se seu mouse possuir o botão de rolagem, mantenha-o pressionado e será possível movimentar a parte ampliada do desenho.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

D DESENHANDO A SEGUNDA PÁGINA DE ESQUEMA ELÉTRICO

D.1. CRIANDO A PÁGINA 2

Exercício 4-1: Criar a página 2 do projeto.

1.CA **Origem**
2.CO **Nova** (na barra **Página**)
Se esta função não estiver ativa, clique no módulo **Diagrama Elétrico** a partir da aba Projeto, e então clique em **Origem** □ **Página** □ **Nova**.
A caixa de diálogo ***Informação da página*** aparecerá.
3.> Descrição da página linha 01
4.# Diagrama de comando
5.> Página
O número 2 é sugerido automaticamente. Não o modifique.
6.> Data de criação da página
A data atual será mostrada nesta linha.
7.> Clique em **OK** para fechar a caixa de diálogo.
O *SEE Electrical* abrirá uma nova página de esquema elétrico.

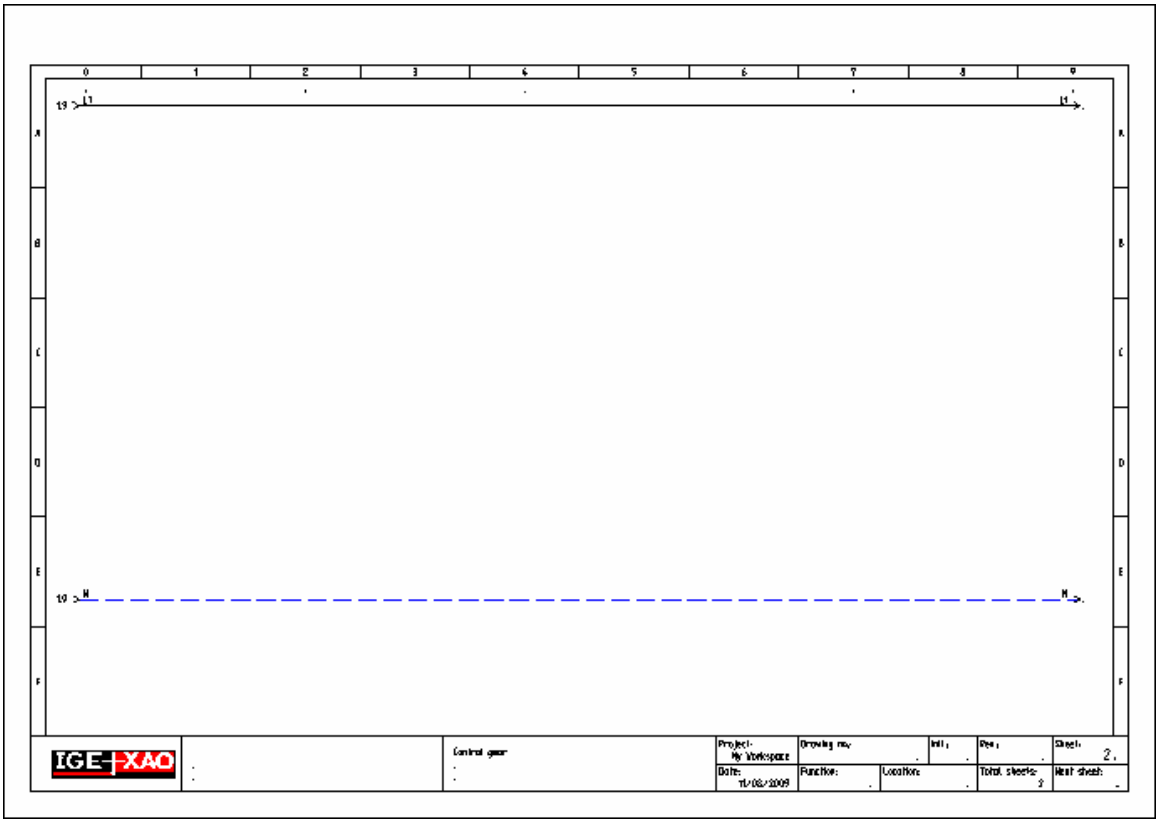
Agora você poderá iniciar o desenho da segunda página de esquema elétrico.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

D.2. PÁGINA 2

Neste capítulo você desenhará os potenciais L1 e N e inserirá components e conexões.

Exercício 4-2: Desenhar o potencial L1.



1.CA **Diagrama Elétrico**
2.CO **Superior** (na barra **Potencial**)
3.> Componente (-)
4.# L1
Digite o nome do potencial.
5.> **OK**
Feche a caixa de diálogo.
A referência cruzada com o mesmo potencial da página 1 aparecerá automaticamente em seu lado esquerdo.

Exercício 4-3: Desenhar o potencial N. Antes, modificar o estilo e cor da pena: “Pontilhado” e azul.

1. A partir da barra **Estilos**, presente na categoria **Desenhar**, clique na seta do ícone .
Selecione "Pontilhado".
2. Na barra **Estilos** da categoria **Desenhar**, clique na seta do ícone .
Selecione "Azul".

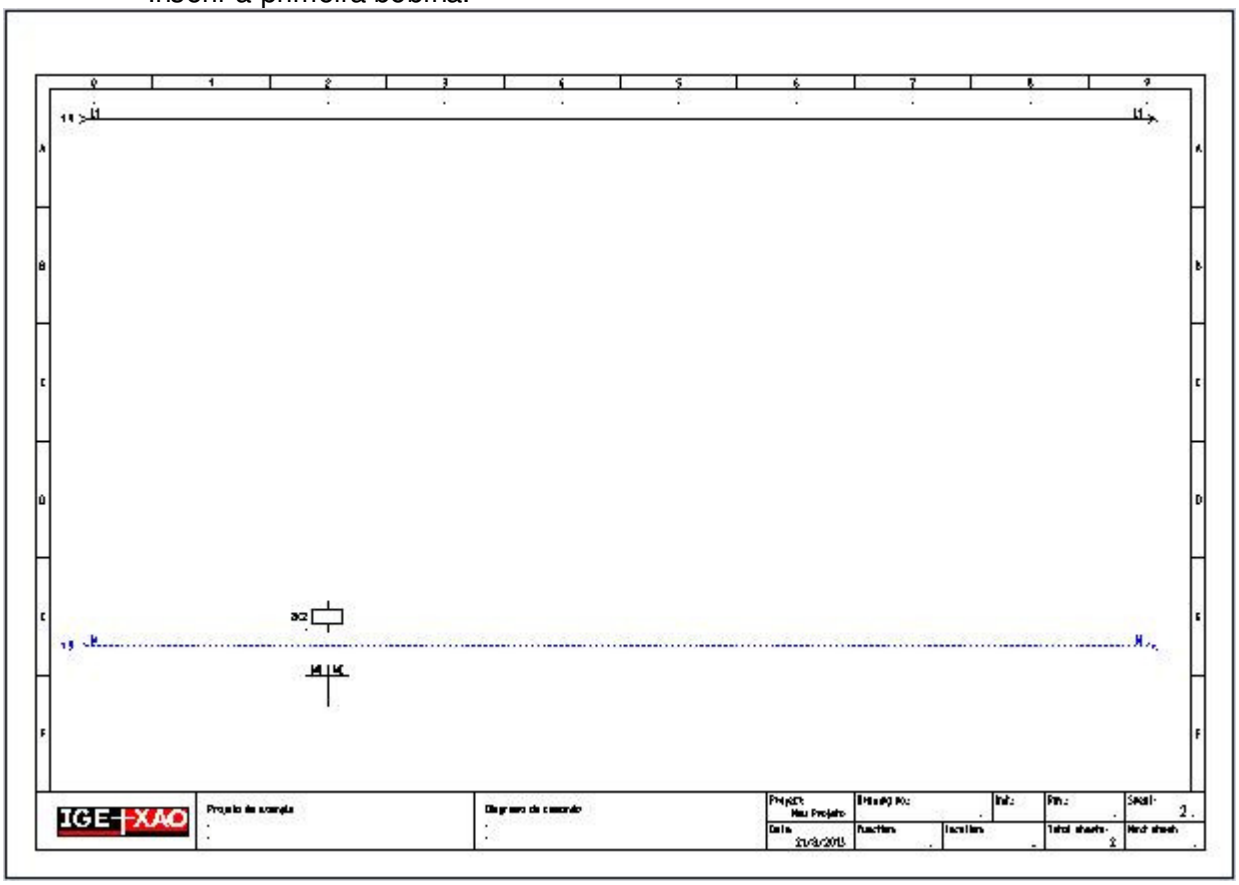
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercício 4-4: Desenhar o potencial N.

1.CA **Diagrama Elétrico**
2.CO **Inferior** (barra **Potencial**)
3.> Componente (-)
4.# N
5.> OK
A caixa de diálogo será fechada.
Modifique o estilo e cor da pena novamente: "Sólida" e preta.
6. Na barra Estilos, modifique para "Sólida".
7. Na barra Estilos, modifique a cor para preta.

Exercício 4-5: Inserir o símbolo da bobina.

- Inserir a primeira bobina.



- Selecionar o banco de dados de símbolos "EN61346-2UK".

1. Movimente o cursor para a aba ***Símbolos***.
Dê um duplo clique no banco de dados de símbolos "*EN61346-2UK*" ou clique no sinal de adição à esquerda do banco de dados de símbolos

.
O banco de dados de símbolos será aberto e suas pastas ficarão visíveis.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

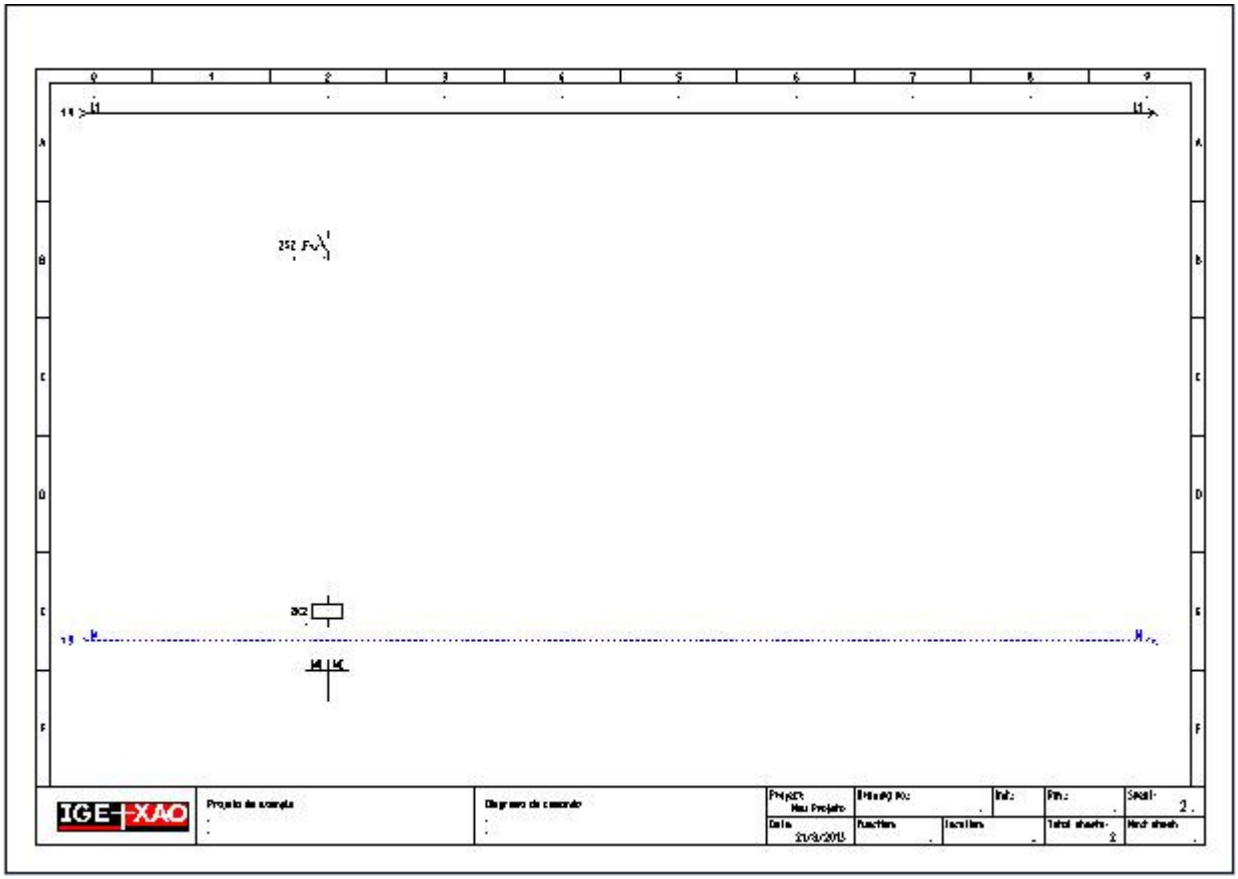
2. Dê um duplo clique na pasta **Relay coils** para abrí-la.
   Se a pasta de símbolos desejada não estiver aberta, use a barra de rolagem para encontrá-la.

3. Selecione o componente 1-pole.
4. Movimente o cursor para o desenho.
   O símbolo aparecerá ligado ao cursor.
5.+ "Solte" o símbolo na posição desejada do desenho na coluna 2.
   A referência em cruz aparecerá em baixo da bobina.

Dica:

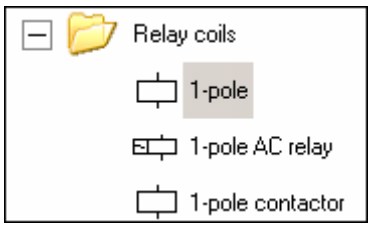
Você poderá mudar a posição da referência em cruz se necessário.

Exercício 4-6: Inserir o símbolo do botão.



1. Movimente o cursor para a guia Símbolos.
2. Feche a pasta "*Relay coils*" clicando no sinal de subtração à esquerda da pasta.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



3. Dê um duplo clique na pasta "*Switchgear, one pole"* para abrí-la.
4. Clique no componente **NO turn detent**.
5. Movimente o cursor para a área de desenho.
   O símbolo aparecerá ligado ao cursor.
6.+ "Solte" o símbolo na posição desejada, na coluna 2.
   Dê um duplo clique no símbolo. A caixa de diálogo ***Propriedades do Componente*** aparecerá.
7.> Conexão 00
8.# 13
   Digite o número.
9.> Conexão 01
10.# 14
   Digite o número.
11.> **OK**
   Feche a caixa de diálogo.

Dica

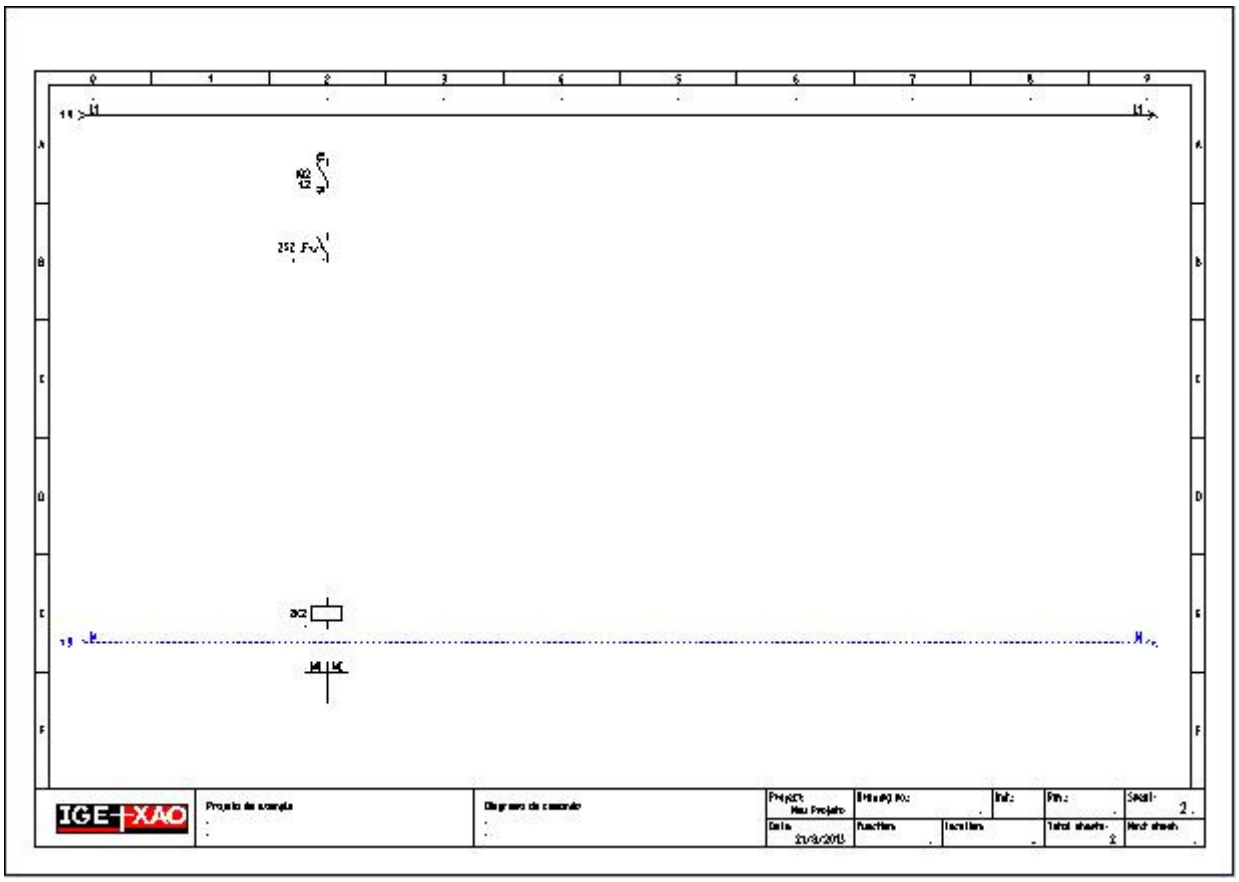
Você pode girar um símbolo 90 graus ou mais antes de inserí-lo.

- *Tecle "+" ou "-" a partir do teclado numérico enquanto o símbolo estiver ligado ao cursor. O símbolo será girado 90 graus no sentido horário ou anti-horário.*

Se você acionar uma destas teclas mais uma vez o símbolo será girado em mais 90 graus.
Você pode fazer o mesmo com as teclas X ou Z..

Exercício 4-7: Inserir um contato NA.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



1. Movimente o cursor para a guia Símbolos.
2. Feche a pasta "*Switchgear one pole*" clicando no sinal de subtração à esquerda do nome.
3. Dê um duplo clique na pasta "*Relay-contacts NO*".
4. Clique no símbolo "1-pole NO"
5. Movimente o cursor para a area de desenho.
   O símbolo aparecerá ligado ao cursor.

6.+ "Solte" o símbolo na posição desejada, na coluna 2.

7.> Componente (-)
   Na sequência será preenchido o nome do componente ao qual o contato pertence.

8.# 1Q2
   Clique no botão Db presente no campo "***Componente***".
   A janela ***Componente função/localização*** aparecerá e nela estarão os componentes com dependência de contatos.
   Selecione o componente a partir da lista.

9.> Conexão 00

10.# 13
   Digite o número.

11.> Conexão 01

12.# 14
   Digite o número.

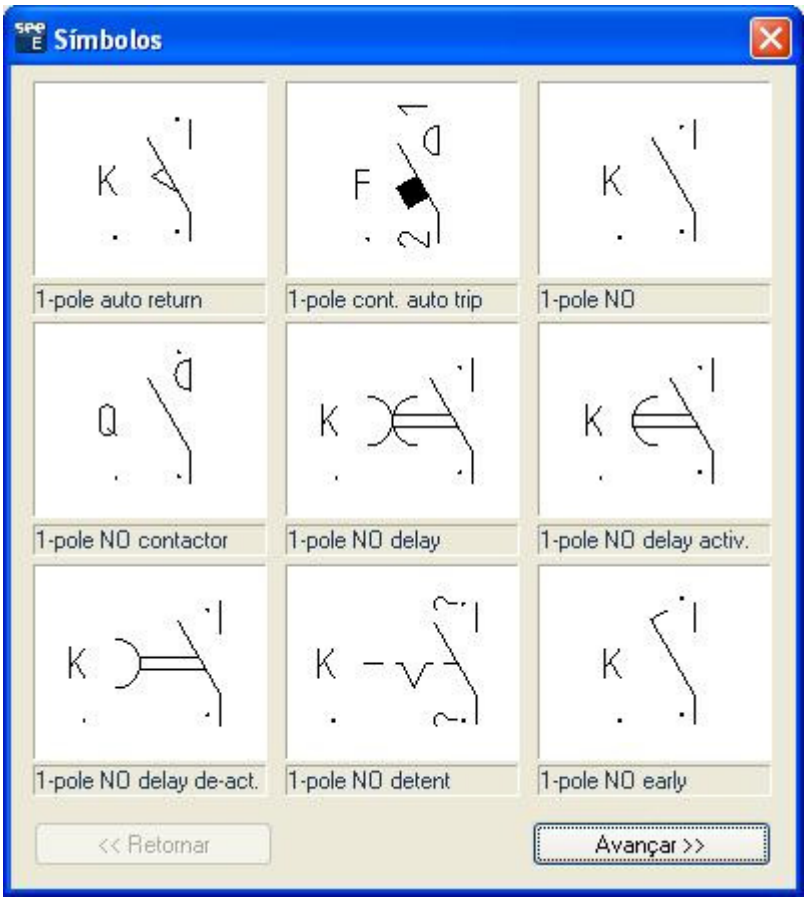
13.> **OK**
   Feche a caixa de diálogo.

Dica:

Você também pode selecionar símbolos a partir da vista gráfica.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

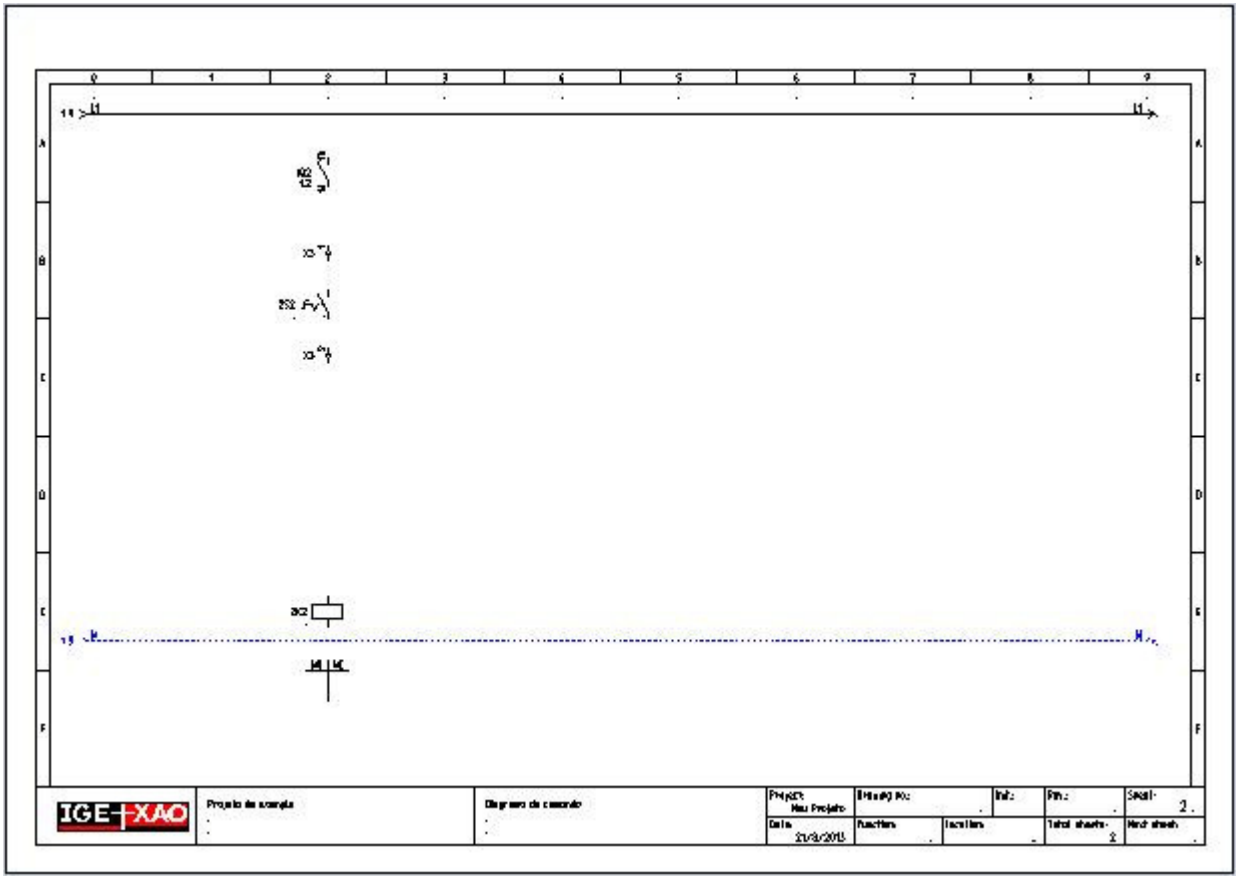
- *Clique com o botão direito na pasta desejada.*
- *Ative o comando* **Vista Gráfica**.



- *Avance ou retroceda entre os símbolos clicando nos botões "**<<Retornar**" e "**Avançar>>**".*
- *Clique no símbolo para selecioná-lo.*

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercício 4-8: Inserir os bornes.



1. Movimente o cursor para a aba Símbolos.
2. Feche a pasta "*Relay-contacts NO*" clicando no sinal de subtração à esquerda do nome da pasta.
3. Dê um duplo clique na pasta "*Terminals*.
4. Clique no símbolo **1 terminal 90° vertical**.
5. Movimente o cursor para a area de desenho.
   O símbolo aparecerá ligado ao cursor.

6.+ "Solte" o símbolo na posição desejada, na coluna 2, entre o contato NA e o botão.
Os dados do componente serão solicitados.

7.> Componente (-)

8.# X3
Digite o nome da régua de bornes.

9.> Número do borne

10.# 1
Preencha o número.

11.> Número de ordem

12.# 1
O número de ordem é usado para classificar os bornes na lista, de forma que components com identificações como PE ou N apareçam no local correto da lista de bornes.

13.> Tipo

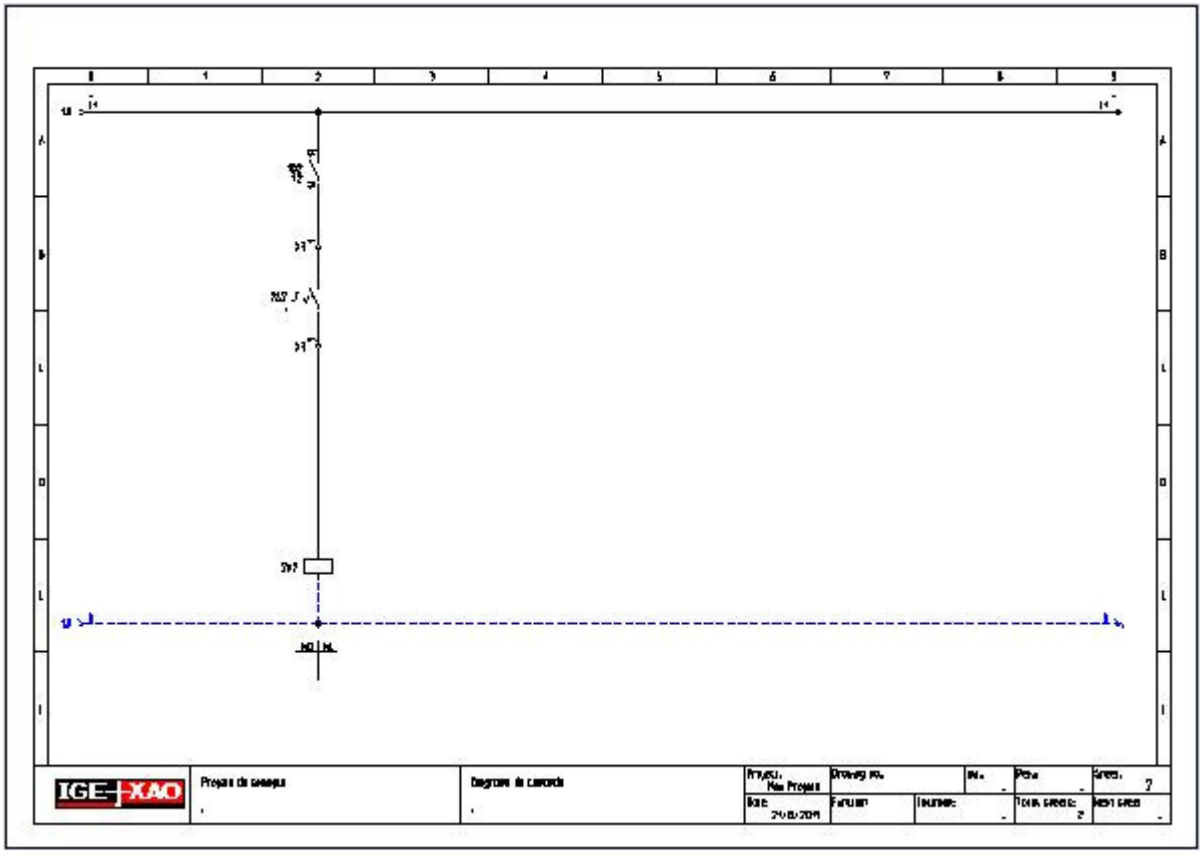
14.# 039061
Digite o tipo do borne.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

15.> **OK**
Feche a caixa de diálogo.

16.+ Insira o próximo borne sob a chave.
O nome do borne será perguntado.
17.> Componente (-)
18.# X3
O nome do componente, número e índice do borne serão sugeridos.
19.> Tipo
20.# 039061
Digite o tipo do borne.
21.> **OK**
A caixa de diálogo será fechada.
Clique com o botão direito do mouse para cancelar o modo de inserção.

Exercício 4-9: Desenhar a conexão.



1.CA **Diagrama elétrico**
2.CO **1 fio** (no painel **Conexão dos fios**)
3.+ Defina o primeiro ponto da conexão no potencial L1, sobre os símbolos.
4.+ Defina o segundo ponto da conexão no potencial N, abaixo dos símbolos.
O fio será desenhado e as conexões serão interrompidas automaticamente onde os símbolos são inseridos.
5. Clique com o botão direito do mouse para sair do modo de desenho.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

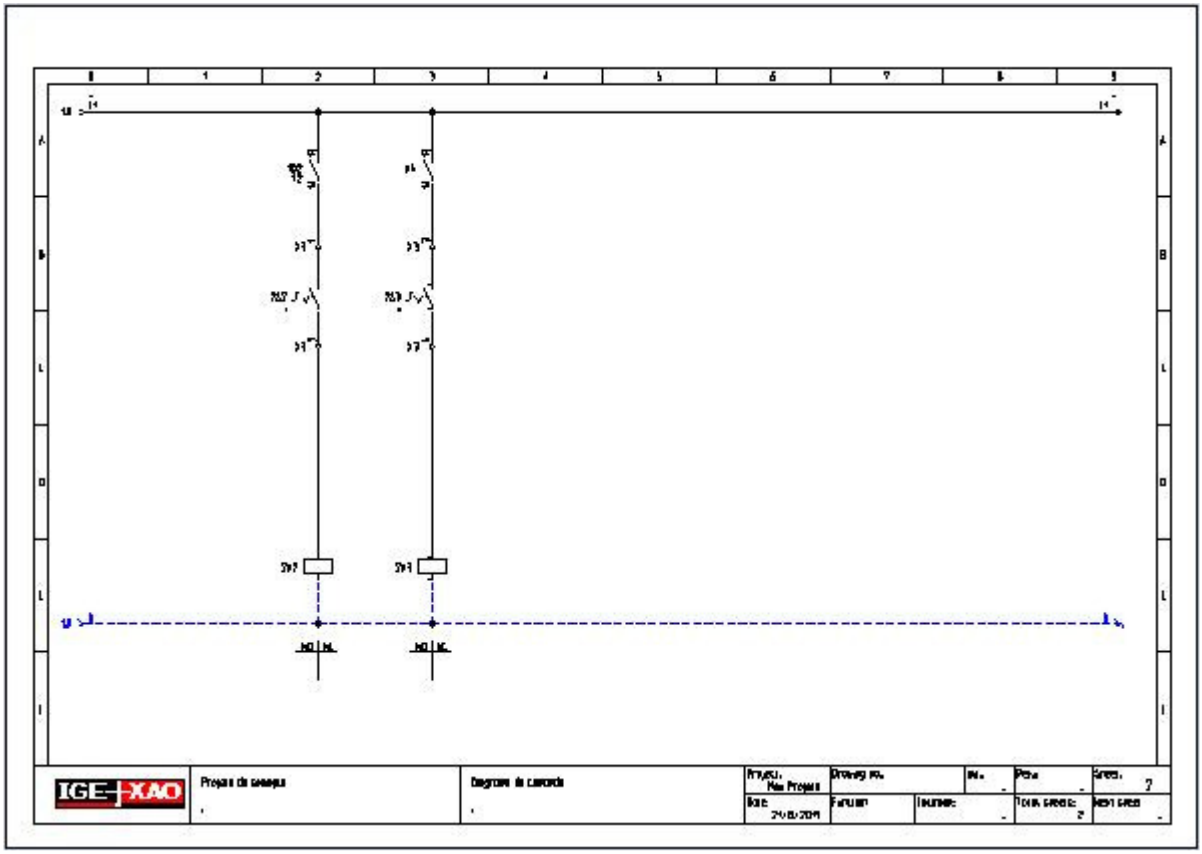
Exercício 4-10: Modificar o estilo e cor da pena para o fio entre a bobina e o potencial N.

1.+ Clique com o botão direito do mouse sobre o fio e selecione o comando **Propriedades** a partir do menu suspenso.
O painel ***Propriedades*** sera mostrado no lado direito da janela do *SEE Electrical*.
2.> Estilo da caneta
3.> Selecione o estilo "Tracejado".
4.> Cord a caneta
5.> Selecione a cor azul.
A cor e estilo da linha serão modificadas dinamicamente.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

D.3. COPIANDO ELEMENTOS EXISTENTES NA PÁGINA 2

Exercício 4-11: Copiar uma coluna do desenho.



1.+ O comando **Geral** □ **Selecionar elementos** □ **Selecionar** estará ativo caso o cursor apareça como uma flecha .

Você pode selecionar um único elemento ou todos os a serem copiados em um retângulo. O retângulo é definido pela seleção de dois pontos opostos.
Neste exercício será usado o retângulo.
Existem duas formas de selecionar parte de um desenho com um retângulo:
Se você quiser selecionar apenas elementos que estejam totalmente dentro da area selecionada, movimente o cursor da esquerda para a direita:
O cursor ficará assim:
Se você quiser selecionar todos os elementos ainda que estejam parcialmente na área selecionada, movimente o cursor da direita para a esquerda:
O cursor ficará assim:

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Dica:

Se você quiser selecionar elementos em desenhos com conteúdo congestionado, pode ser impossível posicionar o primeiro ponto do retângulo e, ao invés disso, será feita a seleção de algum elemento.
A seleção errônea de elementos individuais pode ser evitada conforme segue:

- *Mantenha pressionada a tecla **W** ao clicar o primeiro ponto do retângulo com o botão esquerdo do mouse.*

2.+ Clique e mantenha pressionado o botão esquerdo do mouse para definir o primeiro ponto do retângulo à esquerda, acima do desenho presente na coluna 2.
Os símbolos serão selecionados se estiverem localizados completamente no retângulo.
Arraste o mouse mantendo o botão esquerdo pressionado para definir o segundo ponto do retângulo conforme o passo abaixo.

3.+ Defina o segundo ponto do retângulo na parte inferior direita da coluna desenhada. Todos os componentes e conexões serão destacadas.

4. Após selecionar a coluna, posicione o cursor perto do nó superior. Esse ponto será usado como referência ao fazer a cópia. Mantenha pressionada a tecla CTRL, o botão esquerdo do mouse e movimente-o. Uma cópia da coluna será criada e poderá ser inserida na coluna 3. (Se você não pressionar a tecla CTRL, os componentes selecionados serão apenas movidos)

5.+ Insira a cópia na coluna 3 e solte a tecla CTRL.
A sequência de caixas de diálogo que aparecerão para os components dependerá da sequência em que os components foram inseridos ou deslocados.
O nome do borne sera perguntado.

6.> Componente (-)
X3 será sugerido por padrão. O número e índice deverá ser 3.

7.> **OK**
O nome do contato aberto será solicitado.

8.> Componente (-)
Preencha o nome do contato NA.

9.# 1Q4

10.> **OK**
A caixa de diálogo será fechada.

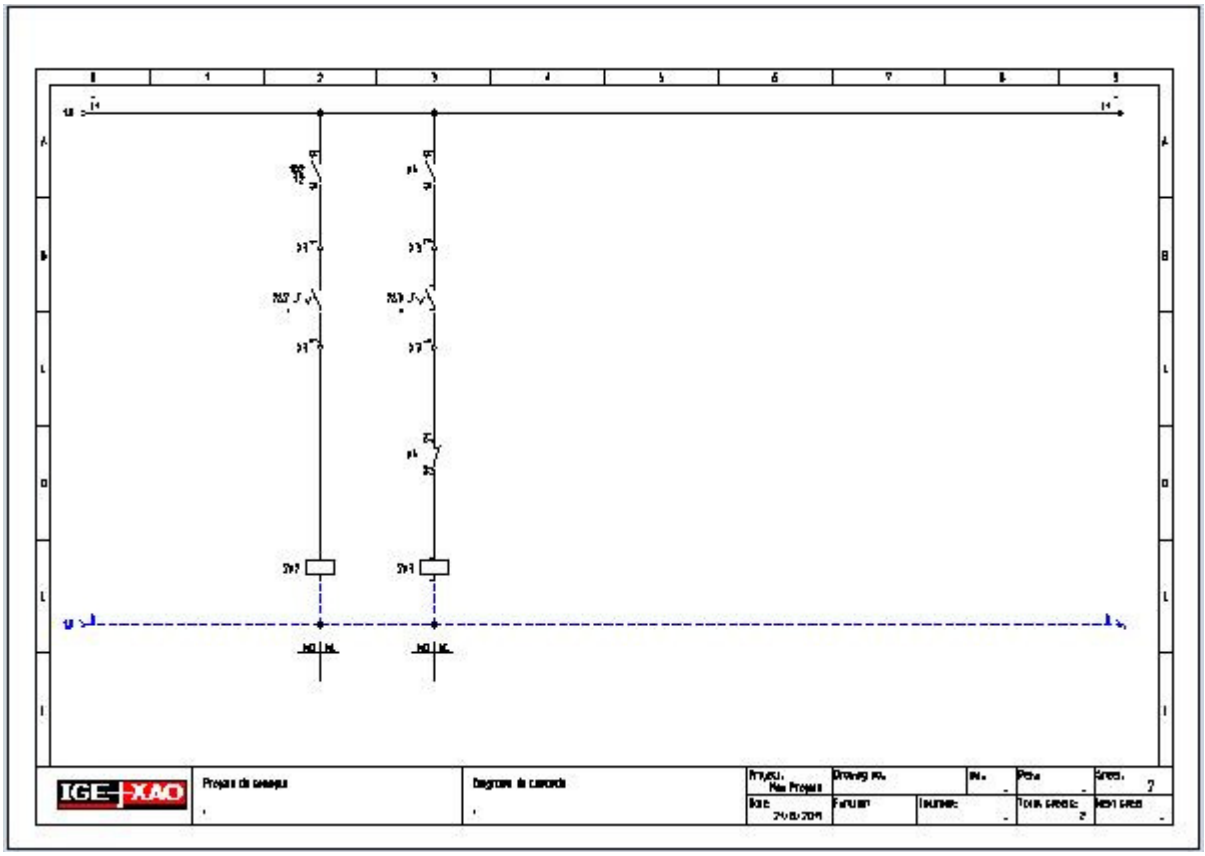
Exercício 4-12: Desfazer a seleção dos componentes.

Você pode desfazer a seleção dos components clicando em um ponto da página em que não existam objetos.

1.CA **Geral**
2.CO **Desfazer** (a partir do menu **Selecionar**)

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercício 4-13: Inserir o símbolo "*1-pole NC*" na conexão existente da coluna 3.



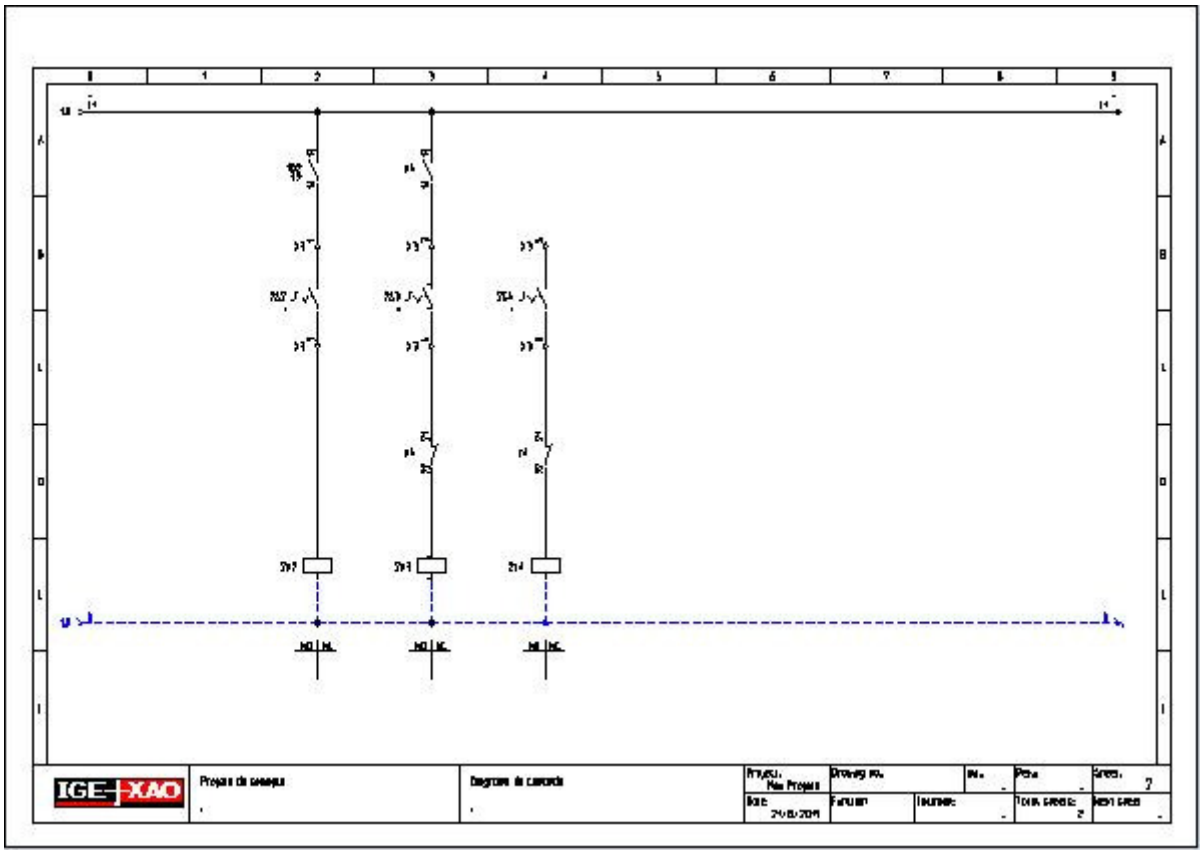
1. Movimente o cursor para a árvore *Símbolos*.
2. Dê um duplo clique na pasta "*Relay-contacts NC*" para abrí-la.
3. Clique no símbolo "*1-pole NC*".
4. Movimente o cursor para a área de desenho.
   O símbolo aparecerá ligado ao cursor.

5.+ "Solte" o símbolo no local sobre a bobina, na coluna 3.
   A conexão será interrompida.
   A caixa de diálogo do contato NF sera mostadada.

6.> Componente (-)
7.# 2K4
8.> Conexão 00
9.# 21
   Insira o número do contato.
10.> Conexão 01
11.# 22
   Insira o número do contato.
12.> **OK**
13. Clique com o botão direito para sair do modo de inserção.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercício 4-14: Copiar os componentes e conexões na coluna 4.



1.+ Defina o primeiro ponto do retângulo de forma que o borne superior esteja dentro dele (na coluna 3).

2.+ Defina o segundo ponto do retângulo abaixo do grupo presente na coluna 3. Todos os componentes dentro do retângulo serão destacados

3. Após selecionar os componentes e fios, clique na conexão superior da bobina. Mantenha pressionada a tecla CTRL, o botão esquerdo do mouse e movimente-o. "Arraste" a cópia do grupo na coluna 4.

4.+ "Solte" a cópia e a tecla CTRL.
As caixas de diálogo para os bornes e o contato aparecerão.
A sequência de caixas de diálogo dependerá da sequência em que os components foram inseridos ou deslocados.
O nome do borne serásolicitado.

5.> Componente (-)
X3 será sugerido por padrão.
Insira o número do borne: 5.
Insira o mesmo valor para índice.
Para os níveis *standard* e superiors do *SEE Electrical,* ao final de **Número do conector** e **Número de ordenação** há o ícone +1. Esta função permite a busca do valor mais alto na régua + 1, sendo a mesma função disponível no índice.

6.> **OK**
O nome do contato NF sera perguntado.

7.> Componente (-) 2K3
Clique no botão Db do campo "***Componente***".

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

A janela ***Componente Função/Localização*** aparecerá com uma lista das bobinas em uso.

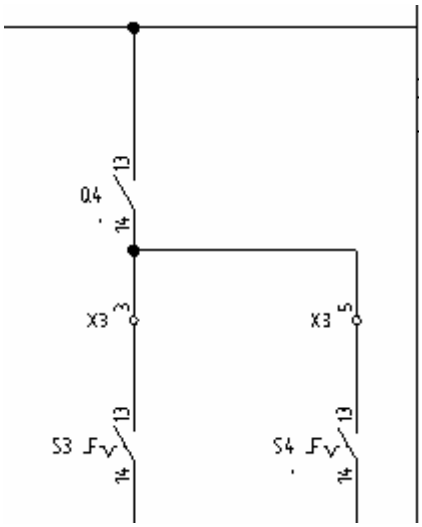
8.> 2K3
Selecione a bobina a partir da lista.

9.> **OK**

Exercício 4-15: Desfazer a seleção dos componente.

1.+ Clique com o botão esquerdo do mouse em algum ponto vazio da tela.

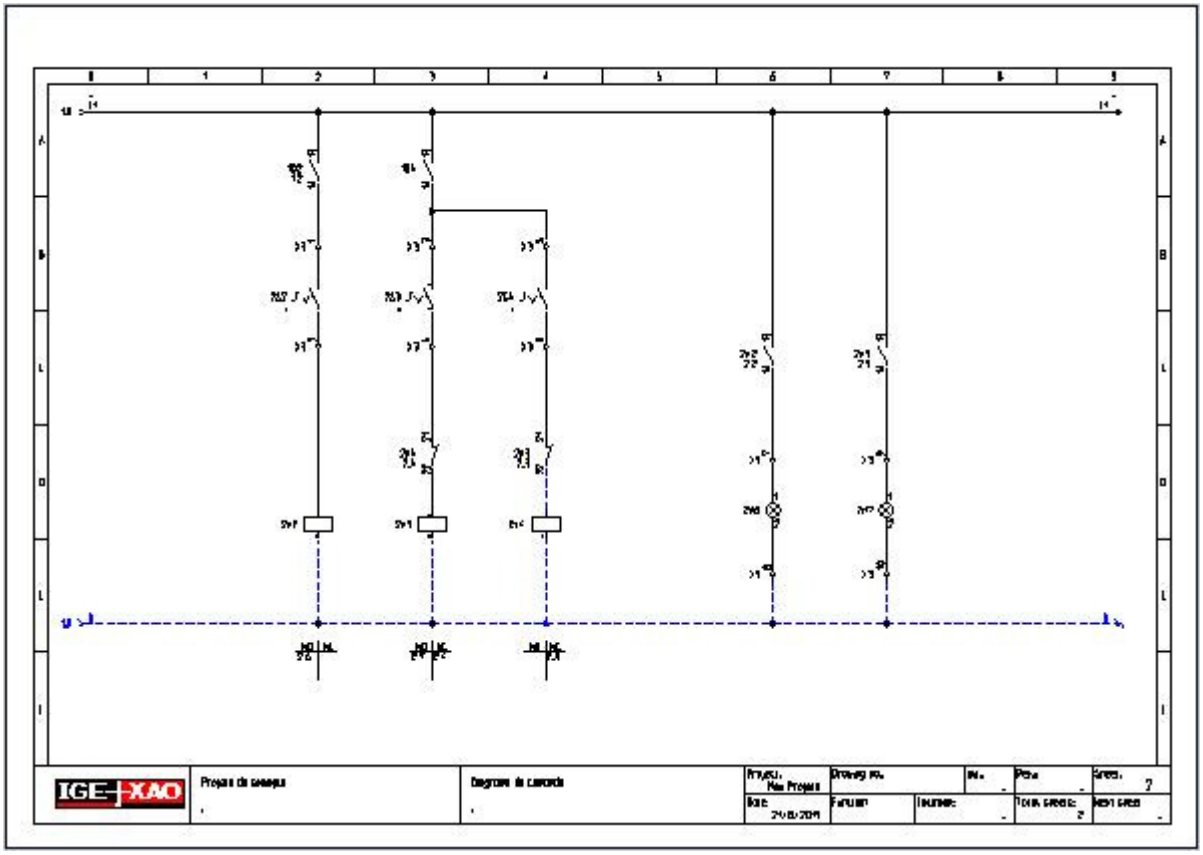
Exercício 4-16: Desenhar a conexão faltante.



1.CA **Diagrama elétrico**
2.CO **1 fio** (a partir do painel **Conexão dos fios**)
3.+ Defina o primeiro ponto na conexão vertical.
4.+ Defina o segundo ponto do fio.
5.+ Defina o ponto final da conexão no borne.
6. Clique com o botão direito do mouse para sair do modo de inserção.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercício 4-17: Desenhar mais uma coluna e copiá-la.



Insira novamente os bornes, um contato NA e a lâmpada a partir do banco de dados de símbolos. A lâmpada está na pasta *Lamps*. Clique com o botão direito do mouse para sair do modo de inserção.

Para o contato NA selecione o nome 2K2 a partir da lista e numere os contatos como 13 e 14. Os bornes devem receber os nomes sugeridos e o tipo 036091.
Desenhe a conexão executando o comando **Diagrama elétrico □ Conexão dos fios □ 1 Fio**. Clique com o botão direito do mouse para sair do modo de inserção.

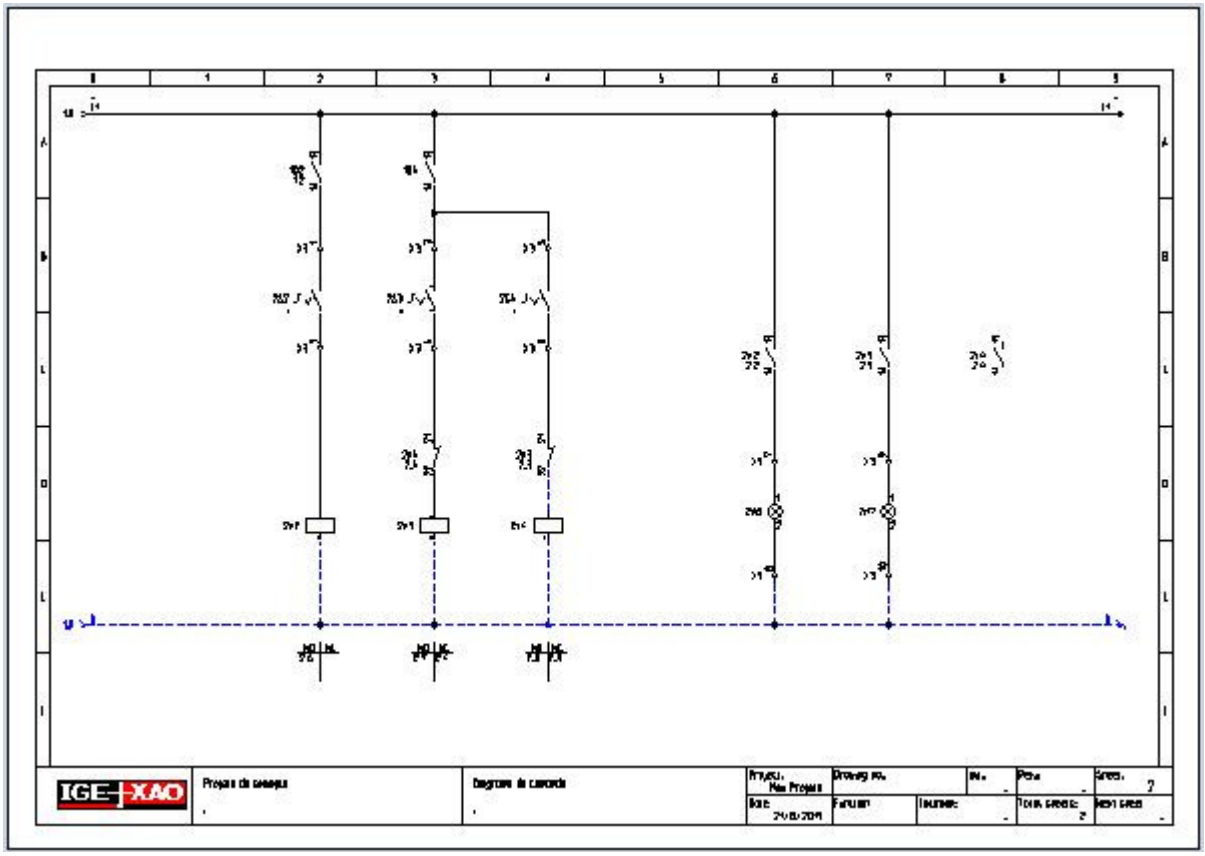
- Selecione o fio entre o borne e o potencial N para mudar o estilo e a cor da pena.

Você poderá copiar o conteúdo desta coluna conforme explicado anteriormente.

- Aceite os nomes sugeridos para os bornes e preencha 2K3 como nome para o contato NA.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercício 4-18: Copie o terceiro contato NA.



1.+ Clique no contato que você deseja copiar. Se o cursor não aparecer como uma flecha, clique em **Geral** □ **Selecionar elementos** □ **Selecionar** para ativar o modo de seleção. O contato sera selecionado.
2. Mantenha a tecla CTRL e o botão esquerdo do mouse pressionados e “arraste” o contato para a coluna 8.
3. "Solte" o símbolo copiado.
A caixa de diálogo do nome do contato aparecerá.
4.> Componente (-)
5.> 2K4
Escolha o nome a partir da janela ***Componente Função/Localização***.
6.> OK
Feche a caixa de diálogo.
7. Clique em qualquer local vazio da área de desenho para desfazer a seleção.

Dica

Você pode inserir múltiplas cópias do objeto selecionado.

- *Selecione o objeto a ser copiado.*
- *Execute o comando* **Copiar**.
- *Execute o comando* **Colar**.

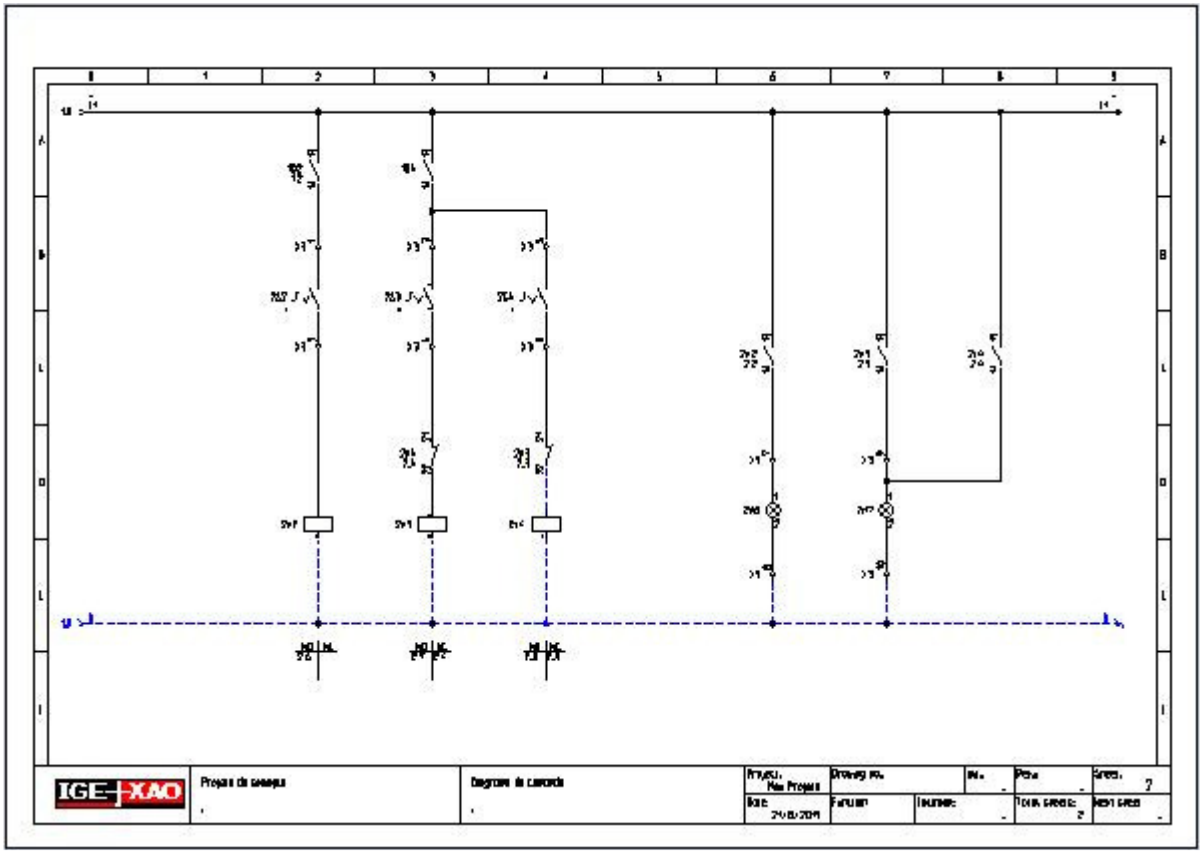
- *Tecle o número correspondente à quantidade de cópias necessárias antes de clicar para posicionar a primeira cópia do objeto.*

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

- *Clique para posicionar a primeira cópia à distância desejada do objeto original. A próxima cópia aparecerá à mesma distância, mas na próxima coluna. O mesmo ocorrerá com as demais cópias.*

Se você digitar 0 será possível criar mais de nove cópias e definir a distância entre elas.
Tenha em mente que esta função não funciona na versão norteamericana.

Exercício 4-19: Desenhar a conexão faltante.



1.CA **Diagrama Elétrico**
2.CO **1 Fio** (a partir do painel **Conexão dos fios**)
3.+ Defina o ponto inicial da conexão a partir do ponto entre o contato NA de 2K3 e o borne.
4.+ Clique para definir o Segundo ponto da conexão.
5.+ Selecione o ponto final do fio no potencial L1.
6. Clique com o botão direito do mouse para sair do modo de desenho.

Exercise 4-21: Salve o projeto.

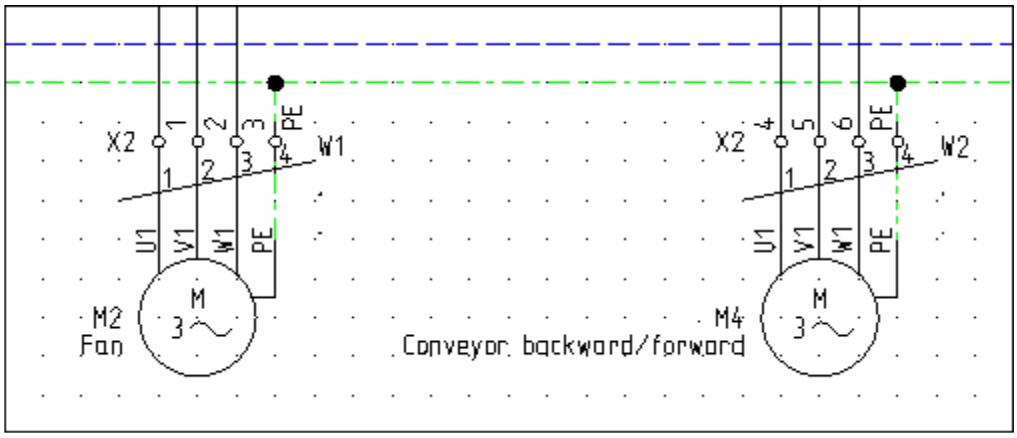
1.CA **Arquivo**
2.CO **Salvar**

Salve seu projeto com frequência. Você também pode clicar no ícone .

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

E DESENHANDO CABOS

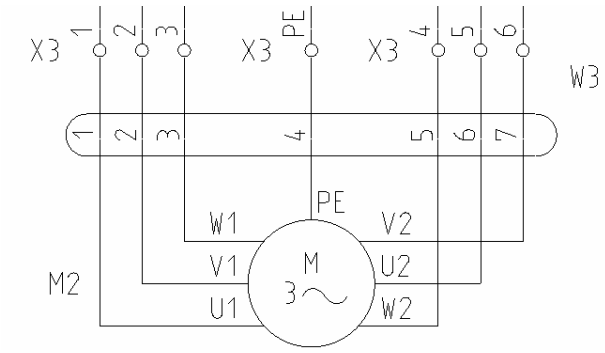
Exercício 5-1: Desenhar os cabos W1 e W2 na página 1.



1.CA **Diagrama Elétrico**
2.CO **Cabo** (no painel **Cabo**)

Nota

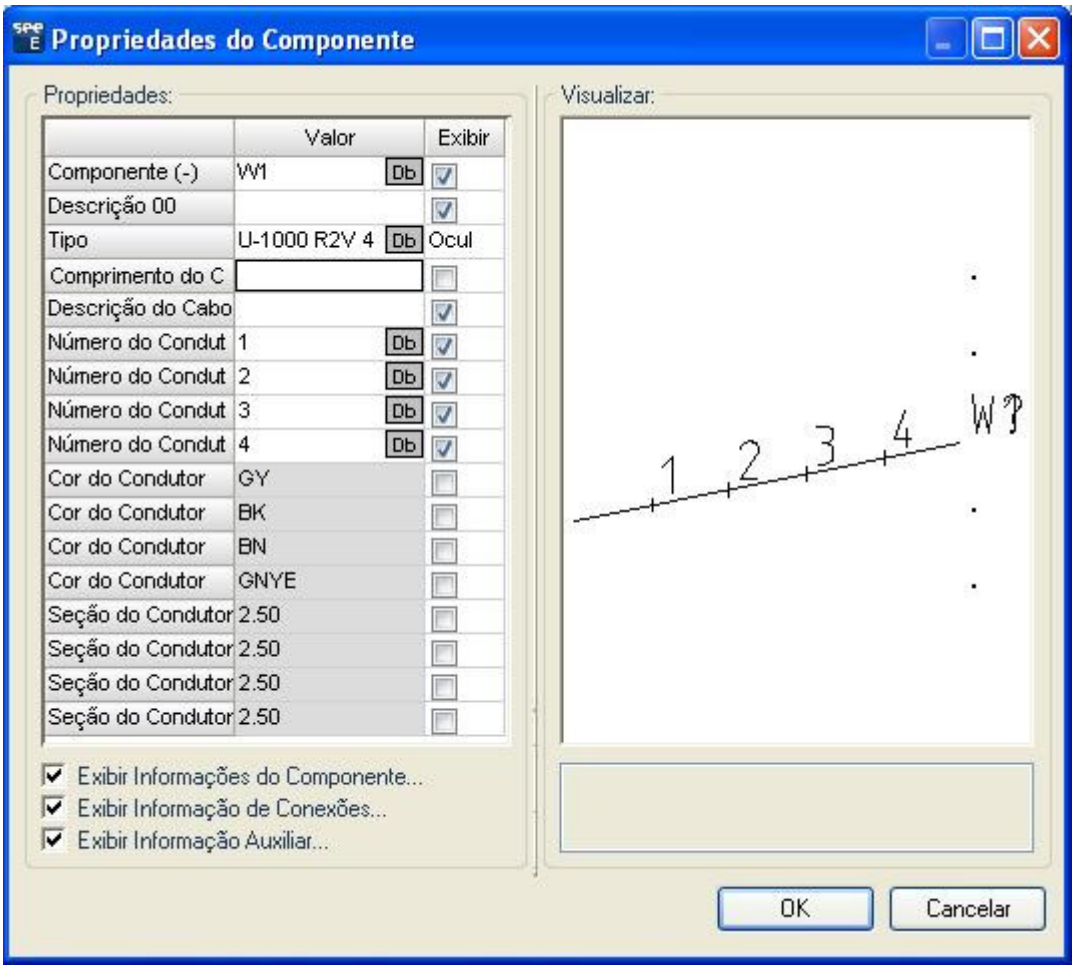
*Caso você tenha selecionado a configuração apropriada na guia **Cabos** da janela **Propriedades de Diagramas Elétricos**, ao executar o comando **Diagrama Elétrico** □ **Cabo** □ **Cabo**, uma caixa de diálogo aparecerá, permitindo inserir um cabo com símbolos previamente definidos para início, meio e fim da parte gráfica. Sua definição é feita a partir do comando **Configurar cabos** a partir da guia **Cabos** presente na janela **Propriedades de Diagramas Elétricos**. Se você selecionar um cabo a partir da caixa de diálogo ele será inserido conforme previamente definido.*
Caso utilize os símbolos para definição de cabos, o do meio será automaticamente prolongado para cobrir qualquer espaço entre os fios.



3.+ Defina o ponto inicial do cabo.
4.+ Defina o ponto final do cabo.
A janela ***Propriedades do componente*** aparecerá.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

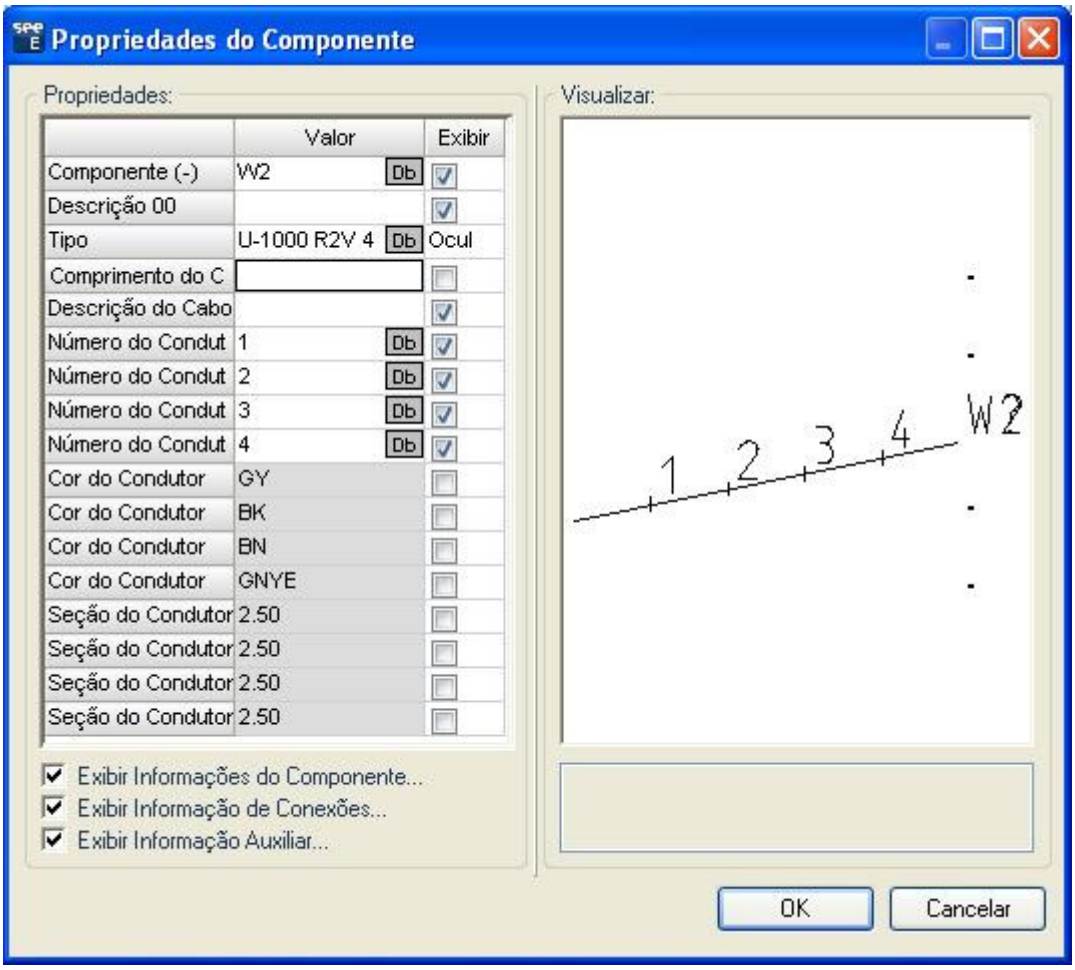
5.# Insira as informações do cabo.
Preencha as informações na caixa de diálogo conforme mostrado abaixo.



6.> Componente (-)
7.# W1
8.> Tipo
9.# U-1000 R2V 4G2,5²
Os números e cores serão preenchidas automaticamente de acordo com o tipo de cabo selecionado.
Ative as caixas de verificação na coluna **Exibir** para definir os dados que serão mostrados no desenho.
10.> **OK**
Desenhe o cabo W2 da mesma forma.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Preencha as informações na caixa de diálogo conforme mostrado abaixo.



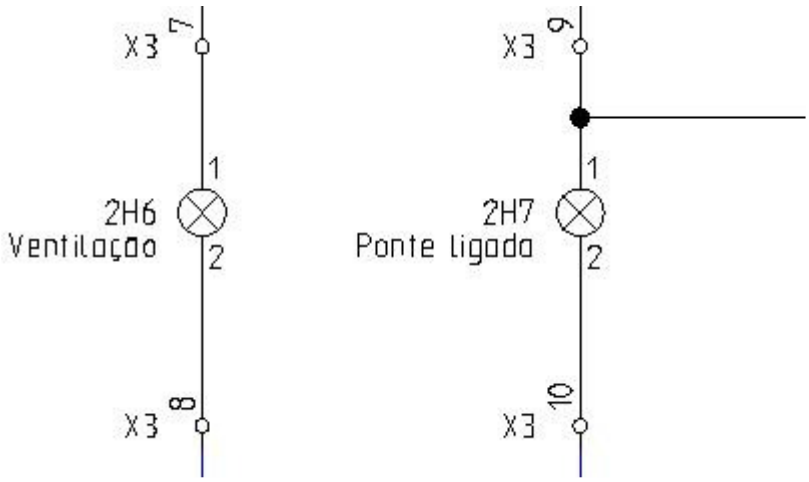
Nota

Se você quiser ligar/desligar a visibilidade dos textos dos condutores do cabo, mantendo a tecla SHIFT pressionada, clique para ativar/desativar a visualização do primeiro texto do cabo. Os textos dos demais condutores serão ligados/desligados.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

F INFORMAÇÕES ADICIONAIS NO ESQUEMA ELÉTRICO

Exercício 6-1: Inserir a descrição nas lâmpadas da página 2.



1.+ Dê um duplo clique na lâmpada da esquerda.
A janela ***Propriedades do componente*** aparecerá.
2.> Descrição
3.# Ventilação
4.> **OK**
A caixa de diálogo será fechada.
5.+ Dê um duplo clique na segunda lâmpada.
6.> Descrição
7.# Ponte ligada
8.> **OK**

Exercício 6-2: Dê nome aos contatos de força.

- Dê um duplo clique na página 1 a partir da guia ***Projeto*** para abrí-la. Você também pode clicar no ícone ou teclar Page Up para ir à página anterior.
- Dê um duplo clique nos contatos de força que estão na coluna 2.

A janela ***Propriedades do componente*** aparecerá.

1.> Componente (-)
2.# 1Q2
Clique no botão Db do campo "***Componente (-)***".
A janela ***Componente Função/Localização*** aparecerá, listando os componentes com dependência de contatos presentes no projeto.
Selecione a bobina 2K2 a partir da lista.
3.> **OK**
A caixa de diálogo será fechada.
Dê um duplo clique nos contatos de força que estão na coluna 4.
A janela ***Propriedades do componente*** aparecerá.
4.> Componente (-)
5.# 2K3

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Clique no botão Db do campo "***Componente (-)***".

A janela ***Componente Função/Localização*** aparecerá, listando os componentes com dependência de contatos presentes no projeto.
Selecione a bobina 2K2 a partir da lista.

6.> **OK**
A caixa de diálogo será fechada.
Dê um duplo clique nos contatos de força da coluna 5.
A janela ***Propriedades do componente*** aparecerá.
7.> Componente (-)
8.# 2K4
Clique no botão Db do campo "***Componente (-)***".
A janela ***Componente Função/Localização*** aparecerá, listando os componentes com dependência de contatos presentes no projeto.
Selecione a bobina 2K4 a partir da lista.
9.> **OK**
A caixa de diálogo será fechada.
A referência cruza será mostrada.

Dica

Você pode navegar entre os components e páginas clicando na referência cruzada dos símbolos. A página correspondente será aberta e o componente destacado com um pino.

Exercício 6-3: Visualizar os potenciais na página 1. Foram criadas referências cruzadas nos poternciais L1 e N para a página 2.

Exercício 6-4: Ir para a página 2. Foram criadas referências cruzadas para a página 1.

1. Para ir a próxima página, clique no ícone ou tecle Page Down a partir do teclado.

Exercício 6-5: Salvar o projeto.

1.CA **Arquivo**
2.CO **Salvar**

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

F.1. USANDO SÍMBOLOS DE REFERÊNCIA CRUZADA

Se um fio não tiver de continuar em outra página, símbolos de referência cruzada deverão ser usados. Os símbolos de referência cruzada estão disponíveis na pasta *References* da biblioteca *EN61346-2UK*. Apenas dois símbolos de referência cruzada podem receber o mesmo nome para que funcionem corretamente. A conexão entre tais símbolos é feita pelo nome do componente.

Exercício 6-6: Inserir dois símbolos de referência cruzada no projeto de exemplo.

- Abra para a página 1.
- Abra a biblioteca de símbolos *EN61346-2UK*. Abra a pasta *References* e selecione o componente "Reference (right)".
- Insira o nome do componente, neste caso, V1.
- Abra a página 2. Selecione o símbolo "Reference (left)" a partir da biblioteca *EN61346-2UK*.
- Insira-o na página e defina novamente o nome como V1.

Em ambos símbolos a referência cruzada sera gerada automaticamente.

É possível inserir símbolos de referencia cruzada ao final de conexões feitas com o comando **Diagrama Elétrico Conexão dos fios 1 fio**.
Para inserir um símbolo de referência cruzada ao desenhar o fio, dê um duplo clique ao final dele. O símbolo de referência cruzada aparecerá automaticamente.

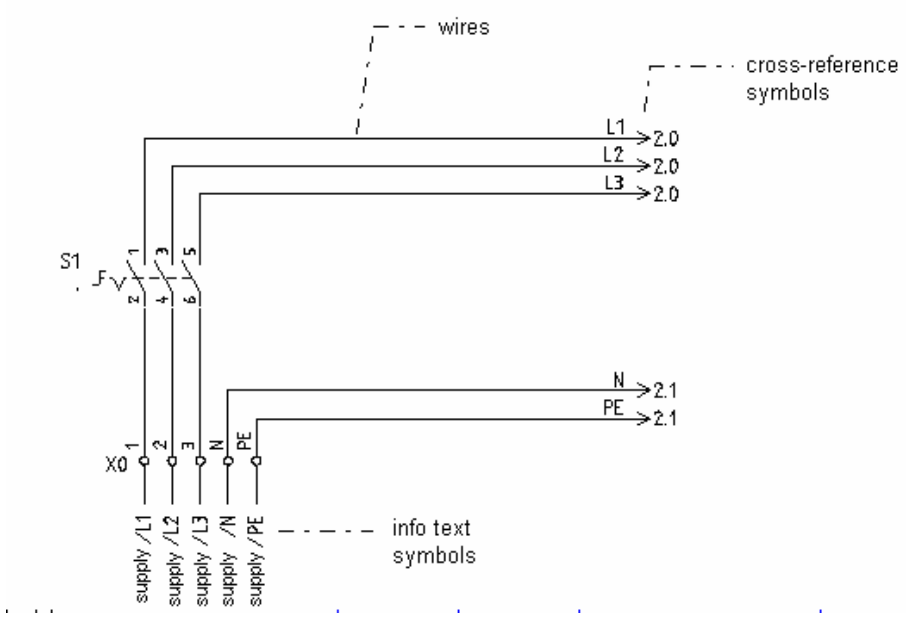
Exercício 6-7: Salvar o projeto.

1.CA **Arquivo**
2.CO **Salvar**

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

F.2. USANDO SÍMBOLOS DE TEXTO DE INFORMAÇÃO

Em determinada situações o uso de símbolos de texto de informação podem ser necessários.



Com o uso de tais símbolos (compostos da parte gráfica, ponto de conexão e nome do componente) ao final do fio tanto ele como o borne terão um destino.
Os textos de informação estão disponíveis na biblioteca *EN61346-2UK*.

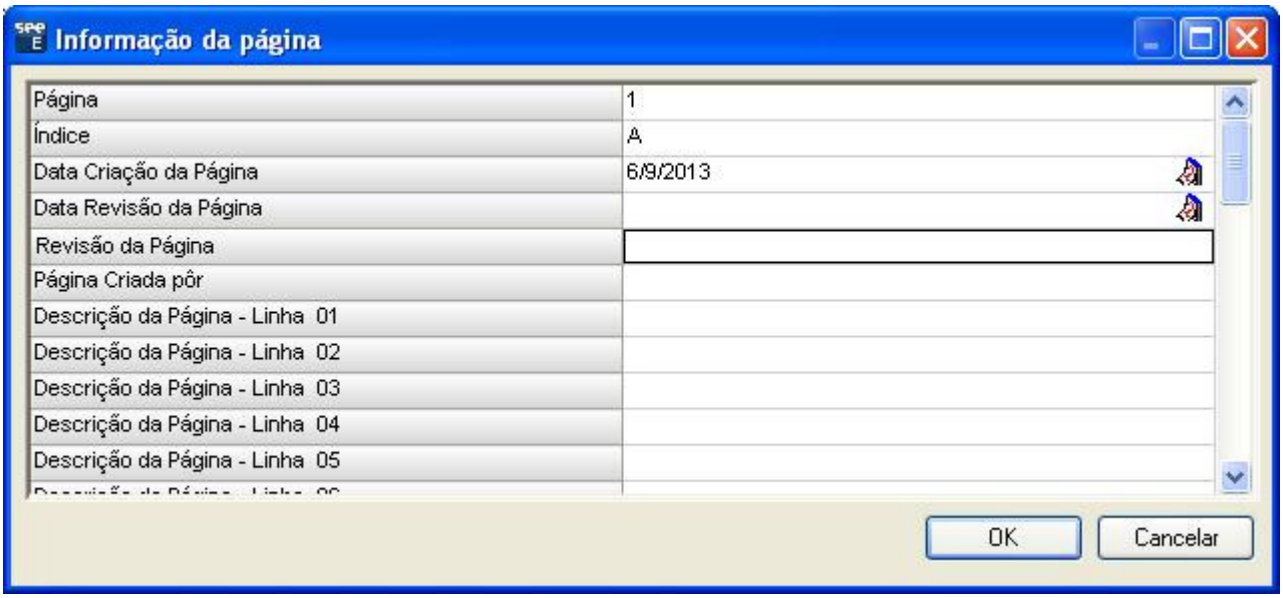
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

F.3. ÍNDICE DA PÁGINA

É comum a necessidade de inserção de páginas entre as já existentes em um projeto por conta das mais variadas situações. Caso a numeração dos componentes esteja vinculada ao número da página, os nomes dos itens já inseridos não podem ser modificados.
O índice da página permite a inserção sem mudança da numeração das já existentes.

Exercício 6-8: Inserir a página 1A no projeto.

- Crie uma nova página usando o procedimento já aplicado.



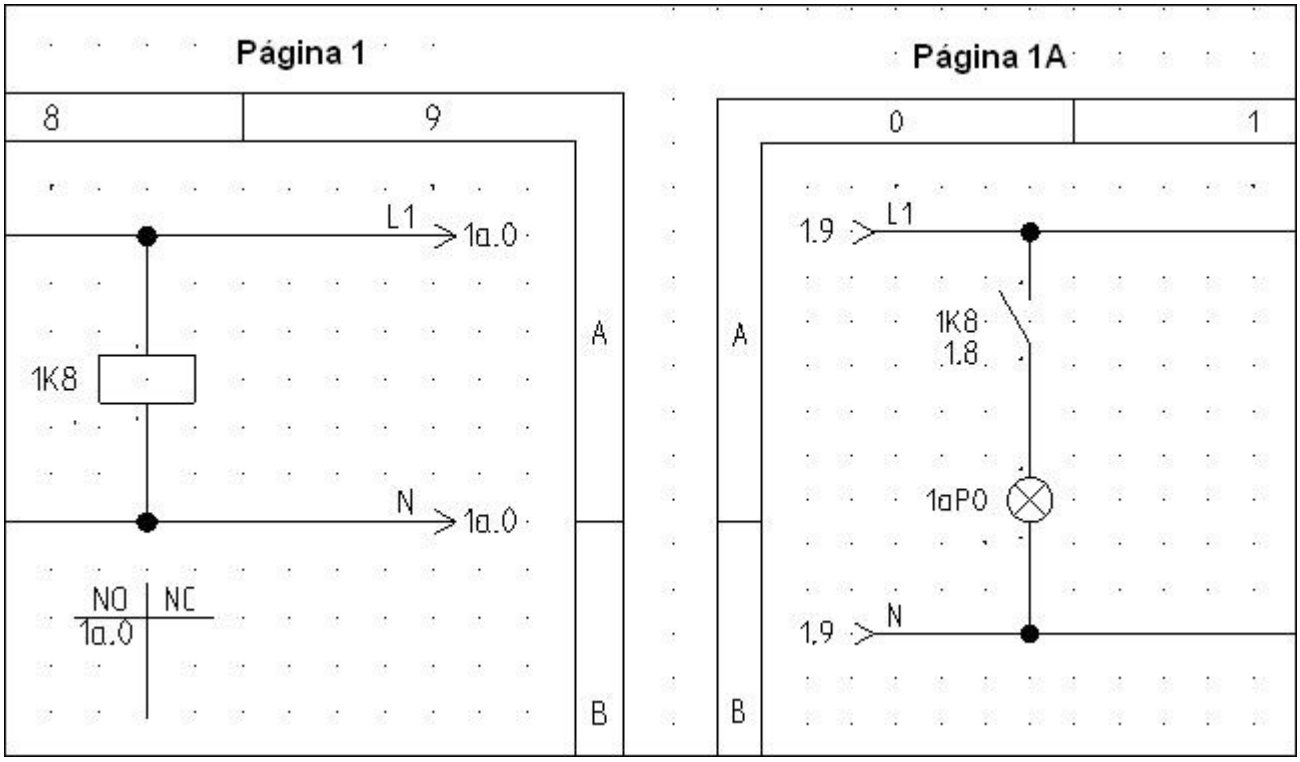
1.> Página
2.# 1
3.> Índice
4.# A
Digite o índice da página.
5.> **OK**

A página será criada.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Dica:

Insira o símbolo de uma lâmpada na página 1A. Se o projeto vincular o nome do componente com o número da página ele será apresentado como 1AP0. Ao desenhar o potencial L1 na página 1A, a referência cruzada nas páginas 1 e 2 serão atualizadas. O mesmo acontecerá com as referências cruzadas nas bobinas e contatos.



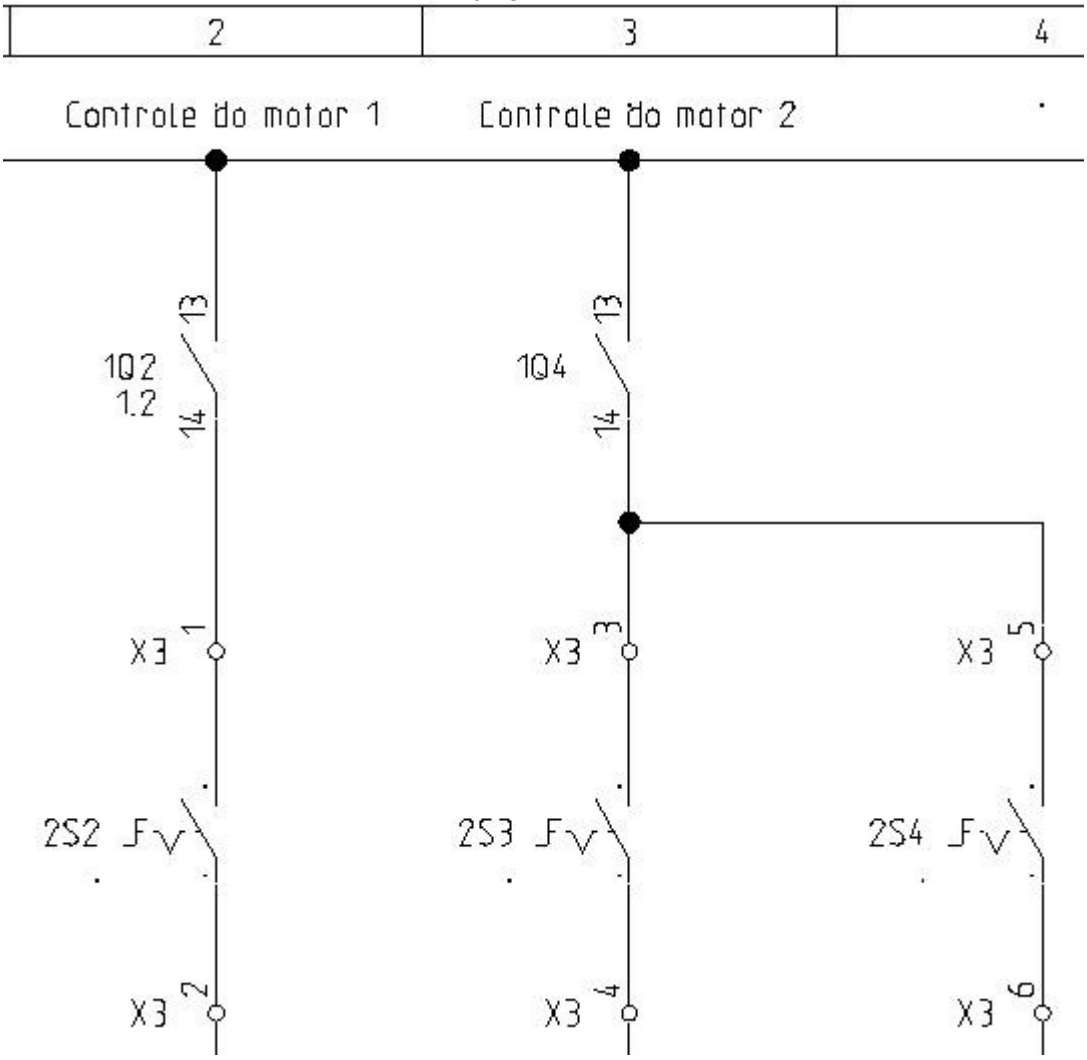
Exercício 6-9: Salvar o projeto.

1.CA **Arquivo**
2.CO **Salvar**

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

F.4. TEXTOS

Exercício 6-10: É possível inserir comentários em uma página. Insira os textos "Controle do motor 1" e "Controle do motor 2" na página 2.



1.CA **Desenhar**
2.CO **Novo Texto** (painel **Elementos**)

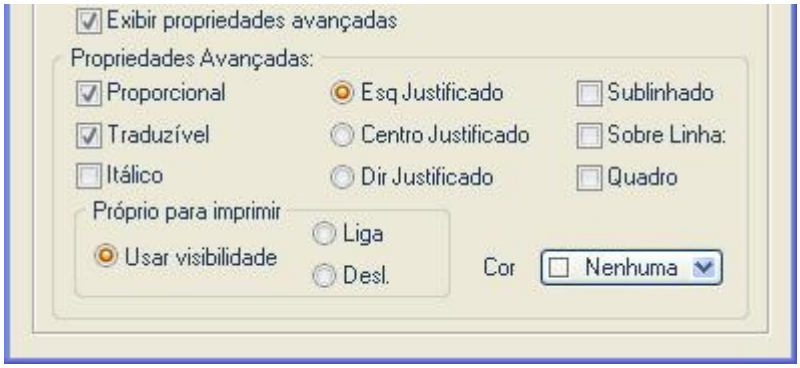
Você também pode clicar no ícone (o ícone permite a criação de um novo texto, enquanto ícone permite a edição de textos existentes).

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

3.+ Movimente o cursor para o campo "***Texto***".

4.# Controle do motor 1
Digite o texto.

5.> Clique na caixa de verificação "Exibir propriedades avançadas" e selecione os atributos de texto necessários, como tamanho, cor e alinhamento (à esquerda, centralizado, direita).



6.+ Insira o texto no desenho clicando na posição desejada.
A caixa de diálogo ***Texto*** continuará aberta.

7.+ Movimente o cursor para o campo "***Texto***" novamente.

8. Modifique o texto existente ou digite um novo, posicionando-o em seguida no desenho.

9.> Clique no botão para fechar a caixa de diálogo ***Texto***.

Exercício 6-11: Modificar o texto inserido anteriormente.

1.CA **Editar**

2.CO **Editar Texto** (painel **Texto**)

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Você também pode clicar no ícone .

3.M Clique no texto que deseja modificar.

4.+ Movimente o cursor para o campo "***Texto***".

5.# <Texto>
Modifique o texto de "Controle do motor 1" para "Motor 1".
A mudança poderá ser vista diretamente no desenho.

6.> Se você quiser, clique na caixa de verificação "***Exibir propriedades avançadas***" e modifique os atributos desejados, como tamanho, cor e alinhamento (esquerda, centralizado ou direita).

7.+ Clique no texto que necessita modificar: "Controle do motor 2" para "Motor 2".
A caixa de diálogo ***Texto*** permanecerá aberta.

8.+ Movimente o cursor para o campo "***Texto***" novamente.

9. Modifique o texto existente.

10.> Clique no botão para fechar a caixa de diálogo ***Texto***.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

G HIPERLIGAÇÕES

G.1. ADMINISTRANDO HYPERLIGAÇÕES

Os comandos a seguir permitem administrar hiperligações (endereços da web ou arquivos) em diferentes páginas do projeto.
Você pode adicionar uma hyperligação a cada objeto presente nos desenhos do *SEE Electrical*.
É possível visualizar quais objetos possuem uma hiperligação por meio do comando **Hiperligação**, a partir da categoria **Editar**.
Se você adicionar uma hiperligação a um símbolo, o link será armazenado na biblioteca juntamente com ele.
Todos os arquivos de imagem (BMP, JPG, TIFF, etc.), arquivos PDF ou do MS Office podem ser vinculados com um objeto.
Ative o comando **Definir** e cole o link do seu navegador (caso seja um endereço da web) ou o caminho para um arquivo que você queira vincular com o objeto.
Você pode usar o comando **Definir** caso queira modificar a hiperligação.
Clique em uma hiperligação e ative o comando **Abrir** para abrir um link.
Caso queira apagar uma hiperligação, clique e ative o comando **Excluir**.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

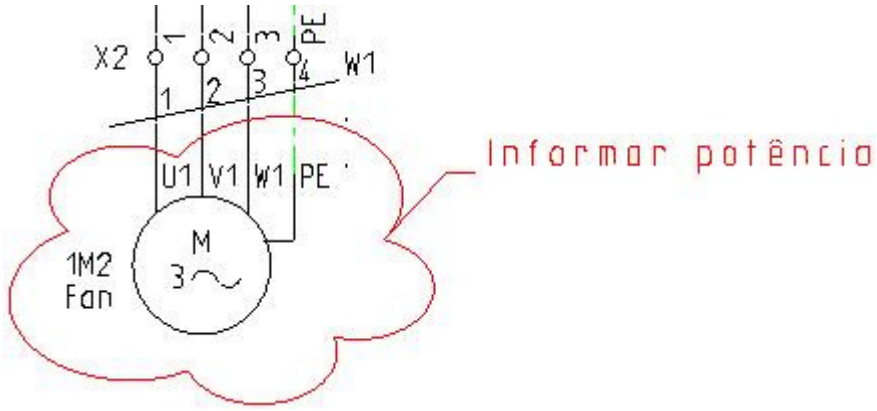
H LEMBRETES

A inserção de lembretes em objetos é feita por meio de marcações gráficas inseridas nos desenhos. Esse comando também está disponível no visualizador do software.
Os elementos marcados são armazenados separadamente no banco de dados de forma que o desenho original não seja modificado ou danificado.
Todos os objetos ficarão envoltos por uma nuvem em vermelho e são salvos no plano 1. Ele consiste da parte gráfica e do texto de lembrete e são processados em uma mesma operação.

Exercício 7-1: Colocar um lembrete no motor da página 1.

- Abra a página 1 do projeto.

1.CA **Legendas e Detalhes**
2.CO **Nuvem**
3.+ Clique o primeiro ponto da nuvem.
A nuvem é definida por dois pontos diagonalmente opostos.
4.+ Clique o segundo ponto da nuvem em um ponto oposto ao primeiro.
5.+ Clique novamente para definir o ponto em que o texto de lembrete será posicionado.
6.+ Digite o texto de lembrete, neste caso "Informar potência".
7# Tecle CTRL+ENTER para iniciar uma nova linha de texto, caso necessário.
8.+ Clique em **OK** para fechar a janela.
9.+ Clique com o botão esquerdo do moute ou tecle ENTER para finalizar o desenho.
A área de lembrete será mostrada:



Clique com o botão direito do mouse para desabilitar a função.

Exercício 7-2: Editar e deslocar o lembrete inserido.

Existem comandos que permitem editar e deslocar lembretes. Comandos convencionais não podem ser usados pois os lembretes também estão disponíveis no visualizador do software.

1.CA **Legendas e Detalhes**
2.CO **Editar texto** (painel **Editar legendas**)
3.+ Clique no texto do lembrete para selecioná-lo.
A janela ***Texto*** aparecerá. Você poderá editar o texto.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

4.+ Clique no botão para aplicar as mudanças.
5.CA **Legendas e Detalhes**
6.CO **Selecionar** (painel **Selecionar legendas**)
7.CO **Mover** (painel **Editar legendas**)
8.+ Clique no lembrete que você deseja deslocar.
9. Clique para definir a nova posição do lembrete.

Exercício 7-3: Selecionar e apagar o lembrete inserido anteriormente.

É possível selecionar e apagar lembretes. Comandos convencionais não podem ser usados pois os lembretes também estão disponíveis no visualizador do software.

1.CA **Legendas e Detalhes**
2.CO **Selecionar** (painel **Selecionar legendas**)
A apar6encia do cursor sera esta .
3.+ Clique no lembrete para selecioná-lo.
Você pode selecionar vários lembretes ao manter pressionadas as teclas CTRL ou SHIFT durante a seleção.
4.CA **Legendas e Detalhes**
5.CO **Apagar legendas** (painel **Selecionar legendas**)
As legendas serão apagadas.
Clique com o botão direito do mouse para cancelar a função.

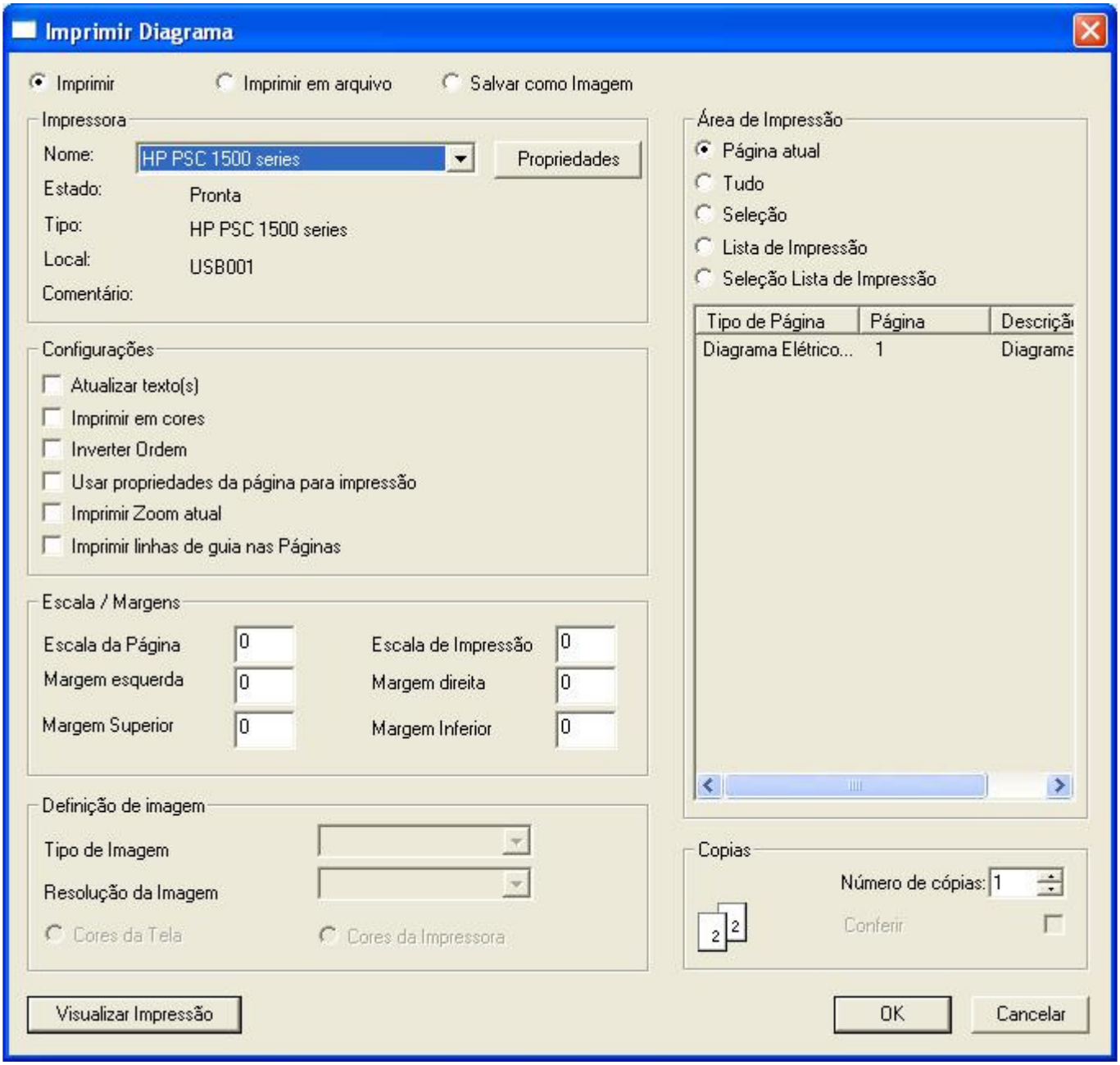
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

I IMPRESSÃO

I.1. IMPRIMIR

Exercício 8-1: Após a conclusão do projeto ele pode ser impresso.

1.CA **Arquivo**
2.CO **Imprimir**
3.CO **Imprimir**

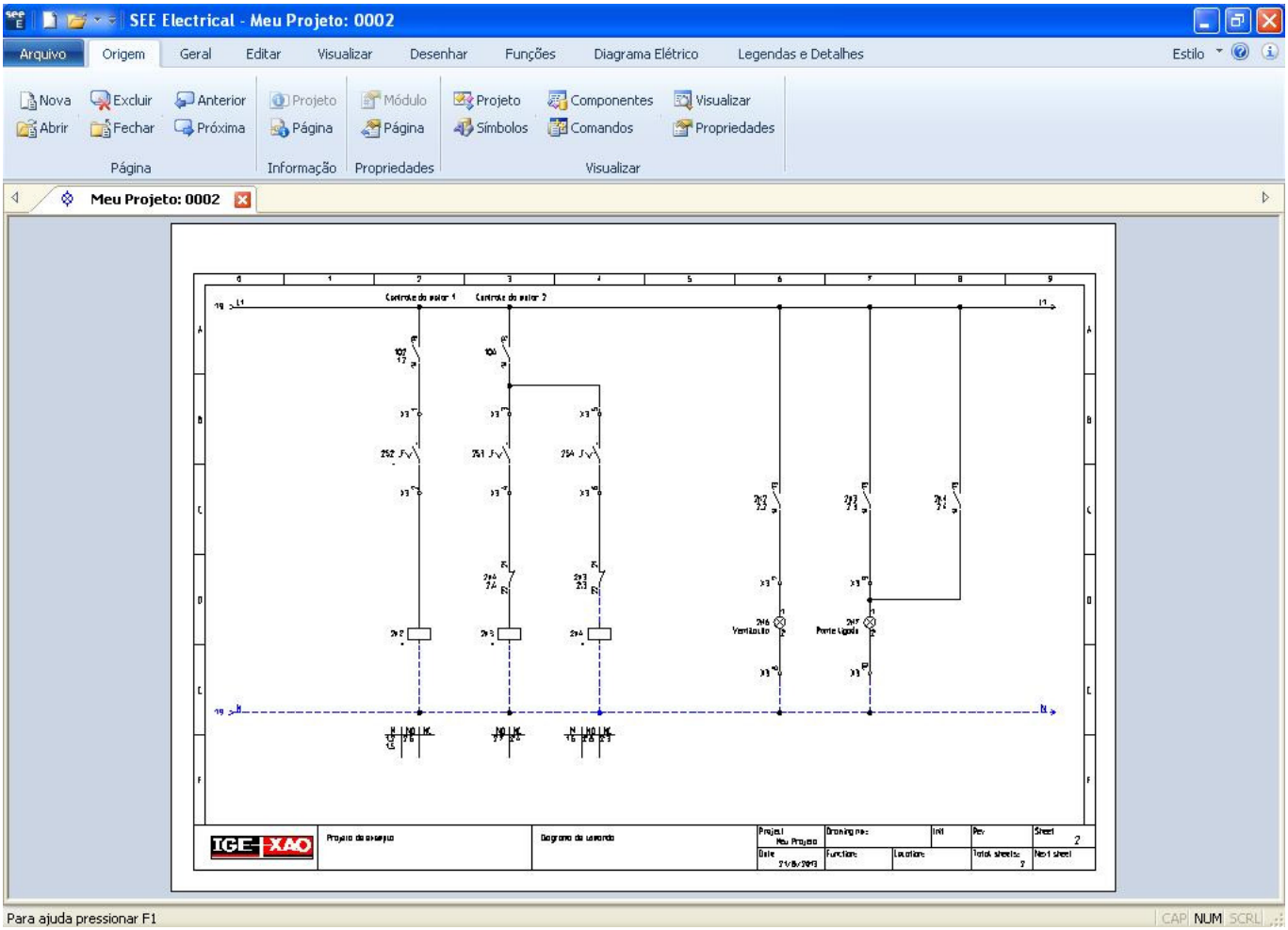


COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

A visualização da impressão pode ser acionada para páginas individuais do projeto. Esta função pode ser acessada clicando no botão **Visualizar Impressão** a partir da caixa de diálogo ***Imprimir*** ou executando o comando **Arquivo** □ **Imprimir** □ **Visualizar impressão**.

Exercício 8-2: Visualizar impressão

1.CA **Arquivo**
2.CO **Imprimir** □ **Visualizar impressão**
A visualização da impressão da página ativa será aberta.



3.> Clique no botão para ampliar a visualização.
4.> Clique no botão **Fechar** para sair do modo de visualização
ou
Tecle ESC.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

J LISTAS DE BANCO DE DADOS

Exercício 9-1: Visualizar as listas de banco de dados do projeto.

- Clique na guia ***Projeto***.

A estrutura em árvore do projeto será aberta. De acordo com o nível (*basic*, *standard*, ou *advanced*), o número de listas disponíveis pode variar.

- A lista a ser visualizada é a de **Componentes**.

1. Componentes
   Dê um duplo clique e a lista será mostrada conforme segue:

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

	Tipo de Documento	Página, Função (=)	Página	Índice	Caminho	Função (=)	Localização (+)	Componente (-)	Descrição 00	Tipo
1	Diagrama Elétrico (EN)		1		0			1S0		S3
2	Diagrama Elétrico (EN)		1		2			1M2	Fan	M10
3	Diagrama Elétrico (EN)		1		2			1Q2	? AMP	3VE1010
4	Diagrama Elétrico (EN)		1		5			1Q5		
5	Diagrama Elétrico (EN)		1		5			1M5		
6	Diagrama Elétrico (EN)		2		2			2S2		
7	Diagrama Elétrico (EN)		2		2			2K2		
8	Diagrama Elétrico (EN)		2		3			2K3		
9	Diagrama Elétrico (EN)		2		3			2S3		
10	Diagrama Elétrico (EN)		2		4			2S4		
11	Diagrama Elétrico (EN)		2		4			2K4		
12	Diagrama Elétrico (EN)		2		6			2H6	Ventilação	
13	Diagrama Elétrico (EN)		2		7			2H7	Ponte ligada	

Exercise 9-2: Filtrar ou classificar as informações da lista.

1.> Clique na coluna "***Componente***"
Clique com o botão direito do mouse.

2.M Selecione **Classificar ordem descendente em linha** a partir do menu suspenso.
Os componentes serão classificados em ordem descendente e os da página 2 serão mostrados antes dos da página 1.

3.> Clique na coluna "Componente" novamente.
Clique com o botão direito do mouse.

4.M Clique em **Ligar filtro -> Componente?** a partir do menu suspenso.

5.# *K*
Digite o valor do filtro.
Caso esteja buscando um componente em específico você pode digitar seu nome completo. Você pode usar os caracteres coringa ? e * para filtrar de acordo com um único caractere (?) ou mais (*).
Ao teclar ENTER, apenas os registros que satisfazem a condição de filtragem serão mostrados.

6.> Clique na guia "Componente" novamente.
Clique com o botão direito do mouse.

7.M Selecione o comando **Remover filtro/classificação** a partir do menu suspenso.
Todos os registros serão mostrados novamente.

Exercise 9-3: É possível salvar as listas em outros programas, como o *MS-Excel* ou *MS-Word*.

1. Abra o *MS-Excel* ou *MS-Word*.

	Tipo de Documento	Página, Função (=)	Página
1	Diagrama Elétrico (EN)		2
2	Diagrama Elétrico (EN)		2
3	Diagrama Elétrico (EN)		2

2..> Clique no canto superior esquerdo da lista de banco de dados conforme mostrado acima.
A lista de banco de dados será selecionada.

3.# CTRL + C
O conteúdo selecionado sera copiado.

4. Cole os dados no *MS-Excel* ou *MS-Word* teclando CTRL + V.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercise 9-4: Salvar o projeto.

1.CA **Arquivo**
2.CO **Salvar**

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

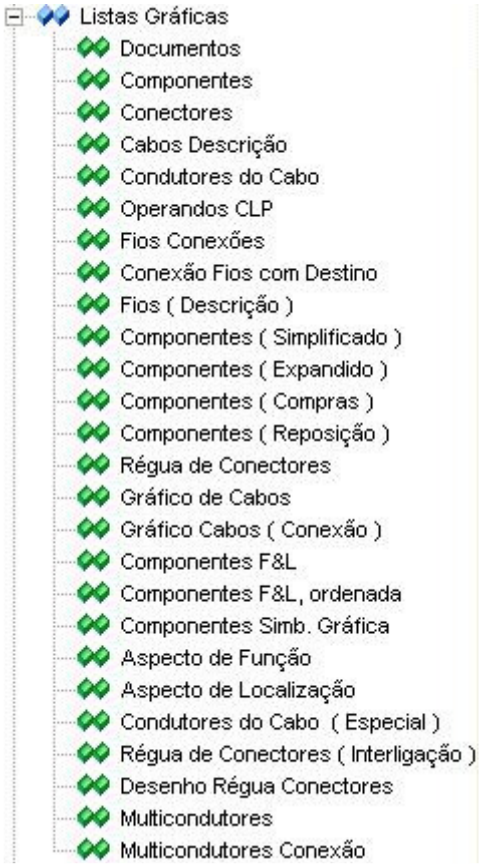
K LISTAS GRÁFICAS

K.1. GERANDO UMA LISTA GRÁFICA

As listas gráficas permitem colocar as informações auxiliares do projeto em formato padrão.

Exercício 10-1: Criar a lista gráfica de componentes do projeto.

1. Expanda o nó *Listas Gráficas* a partir da árvore do *Projeto*.



De acordo com o nível e idioma usado no software, diferentes **Listas gráficas** estarão disponíveis.

2. Selecione a lista gráfica **Componentes** e clique com o botão direito do mouse.
3. CO **Gerar**
   A lista gráfica será gerada.
4. 0001
   Clique no sinal + à esquerda de **Componentes** na árvore de projeto e abra a lista gráfica clicando em 0001.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercício 10-2: Gere as demais listas gráficas usando o mesmo procedimento. Abra-as em seguida.

A lista de bornes e a fiação pode ser gerada no *SEE Electrical standard* e *advanced*.
A régua de bornes, esquema gráfico de bornes e multicondutores pode ser gerada apenas no *SEE Electrical advanced*.
As demais listas de componentes podem ser geradas apenas no *SEE Electrical advanced*.

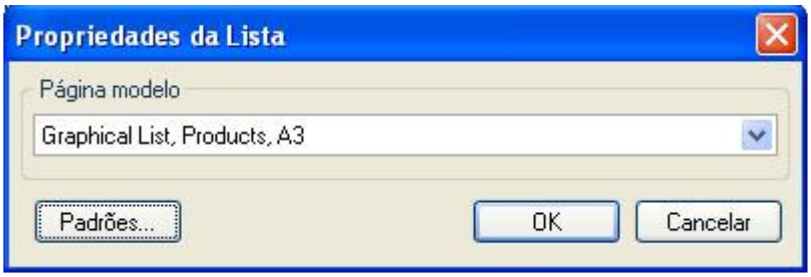
Exercício 10-3: Salvar o projeto.

1.CA **Arquivo**
2.CO **Salvar**

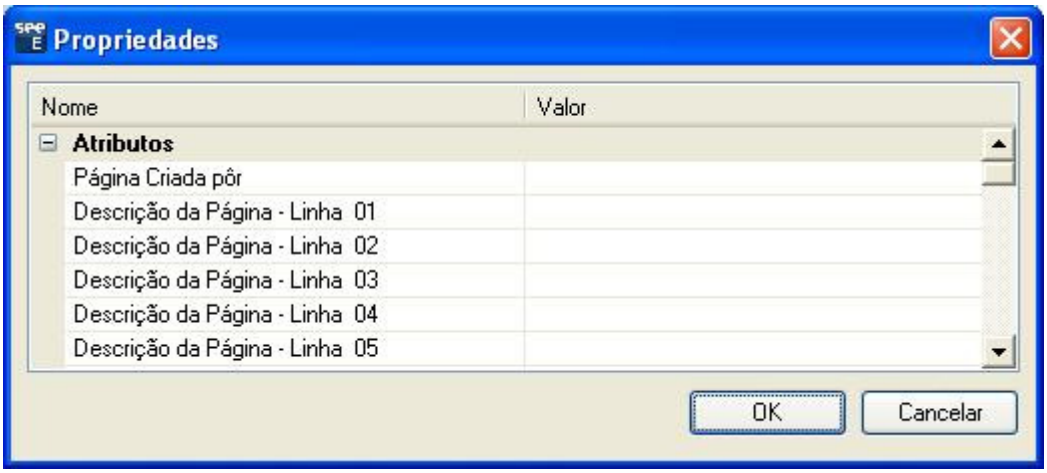
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

K.2. DEFINIR TEXTOS DA PÁGINA PERMANENTEMENTE

O botão **Padrões** está disponível na caixa de diálogo ***Propriedades da lista*** para todas as listas disponíveis.



Você pode definir os textos que aparecerão em função dos atributos do formulário em questão.
Não modifique o texo "Descrição da página – Linha 01" dos formulários de régua de bornes, esquema gráfico de bornes e lista de bornes, pois são preenchidos automaticamente quando de sua geração.



K.3. GERANDO TODAS AS LISTAS GRÁFICAS EM UM ÚNICO PASSO

No *SEE Electrical* você pode gerar todas as listas gráficas em uma única operação.
Nos níveis *Basic* ou *Standard* do *SEE Electrical* você precisará de formulários específicos para a geração de múltiplas listas gráficas em um único passo.
No nível *Advanced* do software há uma ferramenta específica disponível para a geração das listas em um único passo.
Para maiores detalhes quanto a isso, consulte o capítulo **Criando formulários para listas gráficas**.

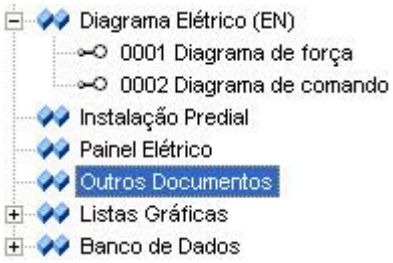
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

L INSERINDO OUTROS DOCUMENTOS

Você pode inserir outros documentos no projeto, como do *Word* ou planilhas do *Excel*. Os tipos de documentos disponíveis dependerão dos programas *Windows* instalados em seu computador que suportem a tecnologia *ActiveX*. Quando você quiser visualizar tal documento o respectivo programa será aberto.

Exercício 11-1: Inserir um documento de outro aplicativo *Windows* que suporte a tecnologia ActiveX:

1. Selecione *Outros documentos* a partir da árvore do projeto.



2.CA **Origem**
3.CO **Novo** (painel **Página**)
O *SEE Electrical* abrirá a caixa de diálogo ***Informação da página***.
4.> Descrição da página - Linha 01
5.# Outras informações
6.> Página
O número 1 será sugerido automaticamente. Não o modifique.
7.> Data de criação da página
A data atual será sugerida.
Você pode preencher outras informações, se necessário.
8.> **OK**
A caixa de diálogo será fechada.
A caixa de diálogo ***Inserir objeto*** aparecerá, permitindo a escolha de outro documento a ser inserido. Nela estão os aplicativos *Windows* instalados em seu computador que suportam a tecnologia *ActiveX*.
9.> Escolha um dos aplicativos mostrados.
10.> **OK**
A caixa de diálogo será fechada. As funções do aplicativo selecionado estarão disponíveis para edição do documento.
11. Edite o documento.

Exercício 11-2: Salvar o projeto.

1.CA **Arquivo**
2.CO **Salvar**

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

M CRIANDO SÍMBOLOS

Os símbolos para diagramas elétricos devem ser criados de acordo com regras rígidas de forma que as conexões sejam reconhecidas e para a correta administração dos componentes utilizados (isso influirá diretamente na respectiva lista).

M.1. CRIANDO UM SÍMBOLO

Tomaremos como exemplo um símbolo composto por um retângulo e quarto retas.

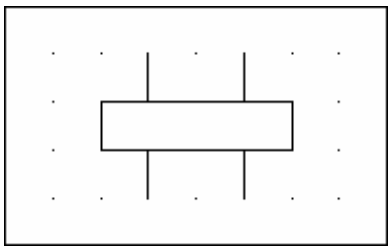
Dicas:

1. *É recomendável o uso de um grid de 5 mm para o desenho de componentes de diagramas elétricos. Você pode desenhar novos componentes usando um grid de 5 mm, 2,5 mm ou menos. Certifique-se de que o símbolo criado se encaixa no grid de 5 mm, especialmente se os pontos de conexão coincidem com ele. Você pode ajustar o grid no painel Estilos. Há também uma função na categoria* ***Visualizar*** *que permite ligar ou desligar a visualização do grid.*

2. *A largura da linha também deve ser controlada. A largura da linha para o desenho de conexões deve ser idêntica da usada no desenho dos fios. A maior parte dos símbolos incluídos nas bibliotecas padrão do software possuem uma largura de linha de 0,25mm.*

Exercício 12-1: Desenhe a parte gráfica do símbolo. Você deve usar funções simples de desenho como Linha, Retângulo, Circunferência, etc. Não use conexões.

- Crie uma nova página em seu projeto.
- Desenhe a parte gráfica.



1.CA **Desenhar**
2.CO **Retângulo** (painel **Elementos**)
Desenhe um retângulo com 20 mm de largura e 5 mm de altura. Isso é possível com o grid padrão de 5mm. A largura e altura do retângulo são mostrados na barra de status, abaixo da área de desenho.
3.+ Clique para definir o canto superior esquerdo do retângulo.
4.+ Clique novamente para definir o canto inferior direito do retângulo.
5. Clique com o botão direito do mouse para sair do modo de desenho.
5. Clique com o botão direito do mouse para sair da função.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

8.CA **Desenhar**
10.CO **Linha** (painel **Elementos**)
Desenhe duas linhas de conexão acima e abaixo do retângulo. Estas linhas devem ter 5 mm de comprimento. O comprimento das linhas é mostrado abaixo da área de desenho.
11.+ Clique o primeiro ponto da linha.
12.+ Clique o segundo ponto da linha.
13. Clique com o botão direito do mouse para encerrar o desenho.
Depois disso, desenhe a segunda linha.
14.+ Clique o primeiro ponto.
15.+ Clique o segundo ponto.
etc.
Clique com o botão direito para finalizar o desenho da última linha.
Clique com o botão direito do mouse para sair do modo de desenho.

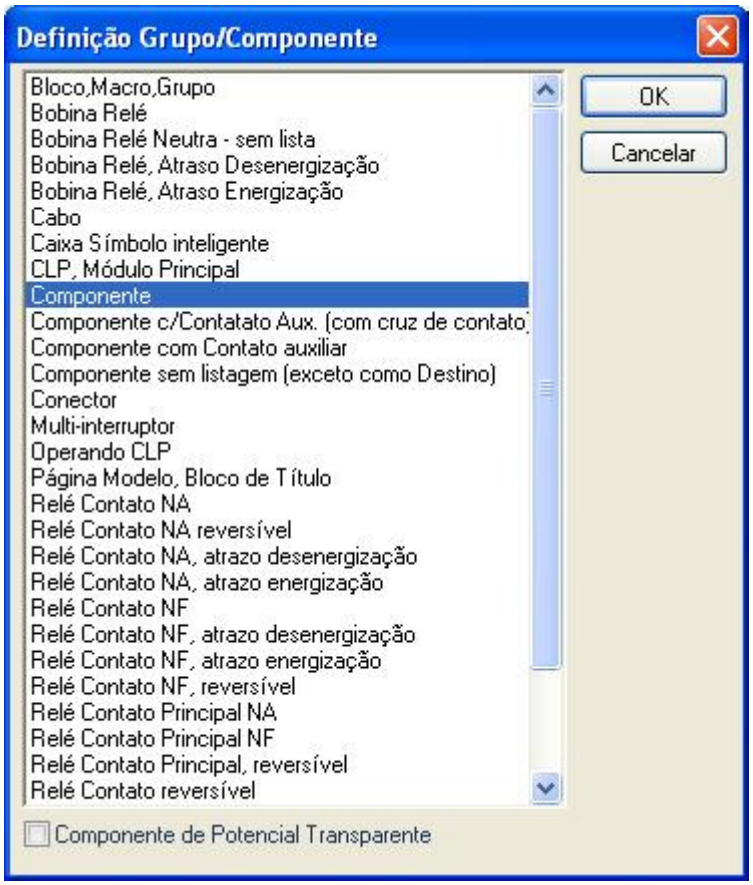
Exercício 12-2: Agrupar o símbolo.

A parte gráfica deve ser definida como símbolo.

1.CA **Geral**
2.CO **Selecionar** (painel **Selecionar elementos**)
3.+ Clique para definir o ponto de onde a seleção começará.
O retângulo deve englobar toda a parte gráfica do símbolo.
4.+ Mantendo o botão esquerdo do mouse pressionado, abra o retângulo e solte-o em seguida.
5. Clique com o botão direito do mouse.
6.M Selecione o comando **Agrupar** a partir do menu suspenso.
A caixa de diálogo ***Definição grupo/componente*** aparecerá.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Selecione o tipo de símbolo, neste caso, Componente.



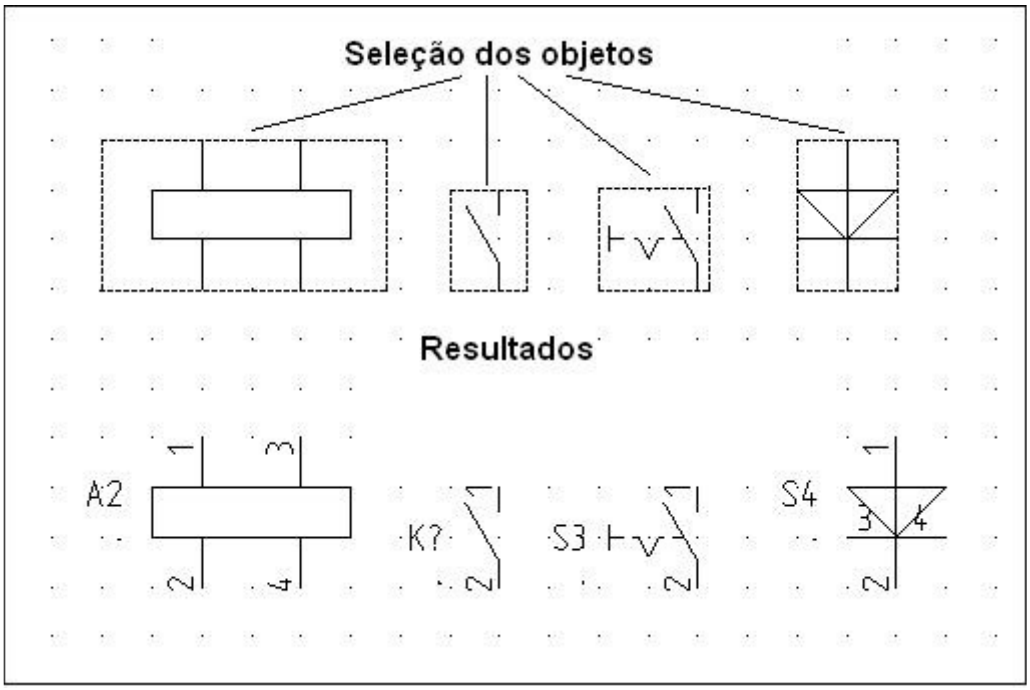
8.> **OK**
O tipo de símbolo selecionado determinará o tipo de lista de banco de dados ou lista gráfica em que ele será mostrado. Uma referência em cruz aparecerá abaixo de um símbolo de **Bobina** mas não de **Componente**.

9.# A
Insira a letra do nome do componente.

10.> **OK**
A parte gráfica está completa e a ela foram adicionados os textos de nome, descrição, tipo e as conexões.
A letra **A** sera usada para composição do nome do componente (**3A8**, por exemplo).

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Dicas para conexões localizadas automaticamente: Os pontos de conexão são inseridos automaticamente ao final de todas as linhas que saiam do símbolo, na horizontal ou vertical.



Em um diodo serão inseridas muitas conexões. Nos exercícios a seguir você aprenderá como apagá-las.
Se um símbolo já possuir conexões, nenhuma outra será inserida de forma automática (caso contrário as conexões em excesso do diodo jamais poderiam ser removidas). Neste caso você também pode copiar as conexões existentes.
A ordem de posicionamento das conexões corresponderá a sequência de linhas criadas (desenhando ou copiando).
Os padrões para textos inseridos automaticamente estão presentes na biblioteca SYSTEM.

Exercício 12-3: Salvar o símbolo em uma biblioteca.

Se o símbolo for usado no futuro é interessante salvá-lo em uma biblioteca. Se este não não for o caso, esse procedimento não é necessário.

1. Ative a guia de símbolos.
   Se a guia ***Projeto*** estiver visível, clique na guia **Símbolos**.
2. MySymbols
   Selecione o banco de dados de símbolos **MySymbols**.
   Você pode salvar símbolos apenas nesta biblioteca ou em outra que tenha criado.
3. Clique com o botão direito do mouse.

4.M **Nova pasta**

5.# Bobina
   Digite o nome da nova pasta de símbolos.

6.CA **Geral**

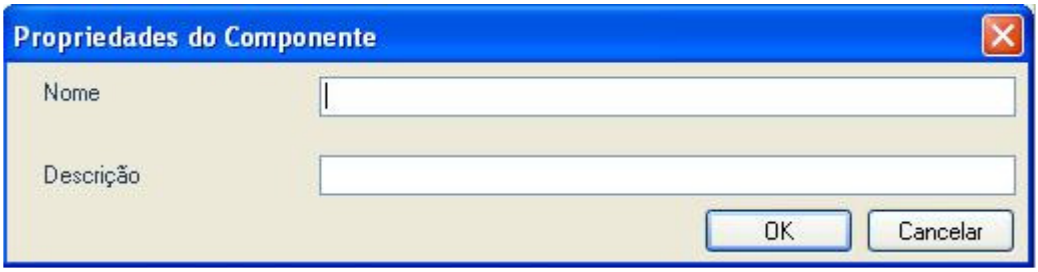
7.CO **Componente** (painel **Selecionar elementos**)

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

8.+ Selecione o símbolo.
9.+ Arraste o símbolo para a nova pasta criada no banco de dados de símbolos. Mantenha pressionado o botão esquerdo do mouse ao arrastá-lo.

Importante!!!!!

Ao arrastar o símbolo, use a conexão superior esquerda como referência. Tal ponto será usado como referência ao inserí-lo no desenho.



10.> Nome
11.# Bobina
Digite o nome e a descrição do símbolo.
12.> Clique em **OK**.
O símbolo será salvo no banco de dados de símbolos.

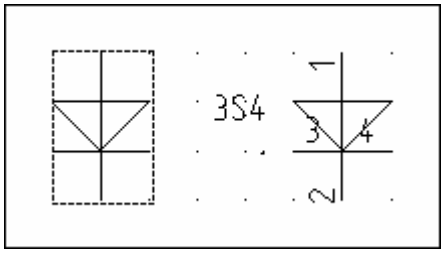
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

M.2. MODIFICANDO SÍMBOLOS EXISTENTES

M.2.1. APAGANDO ELEMENTOS

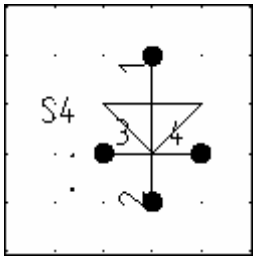
Exercício 12-4: Desenhe um diodo. Agrupe o símbolo como "Componente". Utilize as funções já conhecidas.

O diodo receberá 4 conexões por conta dos extremos das linhas.



Exercício 12-5: Modificar o símbolo do diodo. Primeiramente desagrupe-o.

1.+ Selecione o símbolo.
2. Clique com o botão direito do mouse.
3.CO **Desagrupar**
Execute o commando a partir do menu suspenso.



4. Agora é possível modificar individualmente as partes do componente, como se ele tivesse acabado de ser criado.
Apague as conexões 3 e 4 (as conexões consistem sempre do ponto e texto de conexão. Ao selecionar o texto, o símbolo de conexão será automaticamente selecionado).
5. Agrupe o símbolo novamente como Componente.
Salve o símbolo na pasta *MySymbols*.

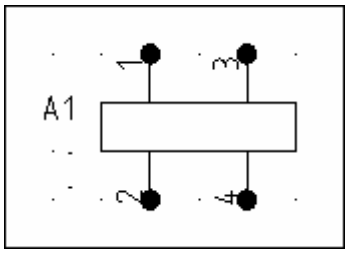
Certifique-se de clicar na conexão superior esquerda ao arrastar o símbolo para a biblioteca.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

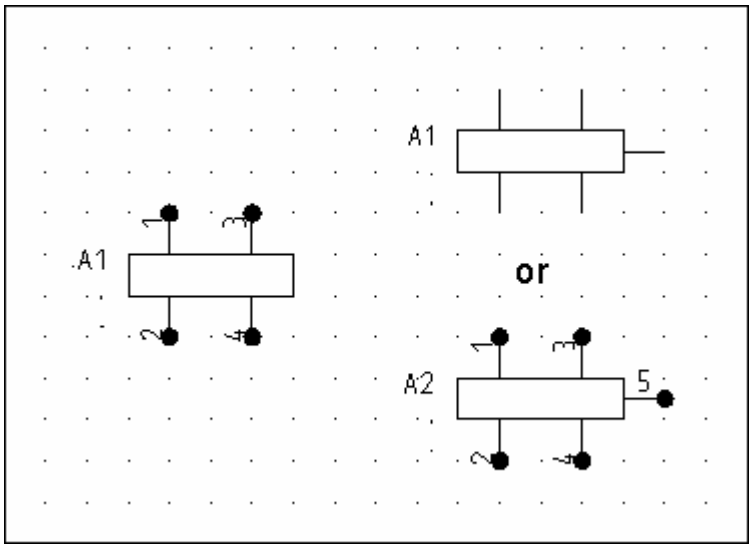
M.2.2. ADICIONANDO ELEMENTOS

Exercício 12-6: Adicionar um ponto de conexão ao símbolo criado no início deste capítulo.

1.+ Selecione o símbolo.
(Caso ele não esteja presente no desenho, insira-o a partir da biblioteca)
2. Clique com o botão direito do mouse.
3.CO **Desagrupar**
Execute o commando a partir do menu suspenso.



4. Desenhe uma linha conforme mostrado anteriormente.
5. Você tanto pode remover todas as conexões disponíveis como copier um texto de conexão existente.



M.2.3. ADICIONANDO TEXTOS

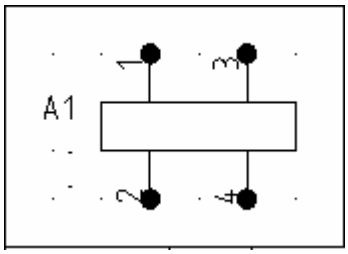
Exercício 12-7: É comum que os textos inseridos automaticamente não sejam suficientes. No símbolo criado no início deste capítulo você pode querer inserir textos descritivos como "Fonte de alimentação" e 12V. Eles não existem no símbolo original

Adicionar os textos ao símbolo criado.

1.+ Selecione o símbolo.
(Se o símbolo não estiver disponível no desenho, insira-o a partir da biblioteca)
2. Clique com o botão direito do mouse.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

3.CO **Desagrupar**
Execute o comando a partir do menu suspenso.



4.CA **Desenhar**
5.CO **Novo texto** (painel **Elementos**)
Os novos textos poderão ser inseridos.

6.> Atributo
Defina o atributo dos textos. Selecione-o a partir da lista.
Nela estão os seguintes grupos de atributos:

- ✓ *Projeto*
- ✓ *Conteúdo*
- ✓ *Função/Localização*
- ✓ *Componente*
- ✓ *Conexão*

e

- ✓ *Outro.*

7.> Componente
Abra o nó **Componente**. Aqui estão textos como nome do componente, índice, descrição e tipo, assim como textos descritivos.

8.> Descrição 01
Dê um duplo clique no atributo **Descrição 01**.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

9.# 12V
Digite o texto.
10.> Alinhado à direita
O texto deve ser alinhado á direita.
11.+ Insira o texto do lado esquerdo do símbolo.
Se a caixa de diálogo **Texto** estiver sobre o local em que você deseja inserir o texto é possível movê-la. Clique com o botão esquerdo do mouse na barra de título da caixa de diálogo e arraste-a para outro local. Solte-a na posição desejada e também o botão esquerdo do mouse.
12. Agrupe o símbolo como Componente novamente.
13. Dê um duplo clique no símbolo.
Você poderá modificar o texto Descrição 01 na caixa de diálogo **Propriedades do componente**.
14. Salve o símbolo na biblioteca MySymbols.
Certifique-se de clicar na conexão superior esquerda ao arrastar o símbolo para o banco de dados.
15. Abra a lista de banco de dados **Componentes**.
Na coluna **Descrição 01** aparecerá o novo texto.
Selecione todos os elementos do símbolo novamente e agrupe-os como **Componente**.

M.2.4. DESLOCANDO TEXTOS

Se um símbolo estiver desagrupado, todos os textos (incluindo os de conexão) podem ser deslocados facilmente usando funções de **Arrastar e Soltar**.

Os textos de conexão e os símbolos de conexão correspondentes formam um conjunto, de forma que são deslocados ao mesmo tempo ao usar as funções de **Arrastar e Soltar**.
Se um símbolo já estiver em uma conexão não é recomendável desagrupá-lo para deslocar o nome do componente, por exemplo.

Neste capítulo, uma função simples para deslocar textos será mostrada, mesmo se o símbolo estiver em uma conexão e sem desagrupá-lo.

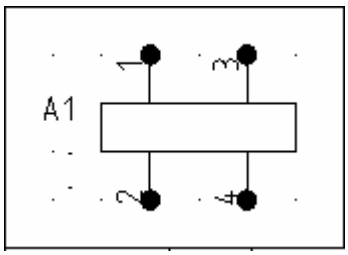
Atenção:

Não *desagrupe o conjunto formato pelo símbolo e o texto de conexão com o comando* **Desagrupar***!!!*

Exercício 12-8: Deslocar o nome do componente e os textos de conexão. Desagrupe o símbolo antes.

1.+ Selecione o símbolo.
(Se o símbolo não estiver disponível no desenho, insira-o a partir da biblioteca)
2. Clique com o botão direito do mouse.
3.CO **Desagrupar**
Execute o commando a partir do menu suspenso.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



4.+ Deslocar o nome do componente.
Para fazer isso utilize o recurso de **Arrastar e Soltar**. Clique no texto, mantenha o botão esquerdo do mouse pressionado e desloque-o para a posição desejada. Depois disso, solte o botão esquerdo do mouse.
5.+ Deslocar um texto de conexão.
6.CA **Geral**
7.CO **Parcial** (painel **Selecionar elementos**)
O símbolo e o texto de conexão fazem parte de um conjunto. Com a função **Parcial** é possível selecionar o texto ou o símbolo separadamente.
8.+ Selecione um texto de conexão.
9 M. Clique com o botão direito do mouse e ative o comando **Mover** a partir do menu suspenso.
10.+ Defina o ponto de referência para o deslocamento do texto de conexão.
11.+ Posicione o texto no local desejado.
Faça o mesmo com os demais textos de conexão.
Agrupe o conteúdo como "*Componente*" novamente.

Exercício 12-9: Deslocar o nome do componente.

- Não desagrupe o símbolo!

1.CA **Geral**
2.CO **Parcial** (painel **Selecionar elementos**)
3.+ Selecione o nome do componente.
4.M Clique com o botão direito do mouse e acione o comando **Mover** a partir do menu suspenso.
5.+ Defina o ponto de referência para o deslocamento do texto.
6.+ Posicione o texto no local desejado.
Com este comando você pode deslocar todos os textos sem desagrupar o componente.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercício 12-10: Gire os textos dos números de conexão.

- Não desagrupe o símbolo!

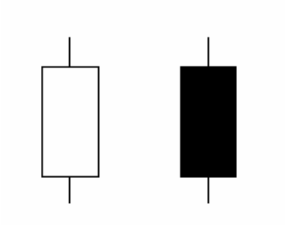
1.CA **Editar**
2.CO **Editar texto** (a partir do painel **Texto**)
3.+ Clique no texto que você deseja girar.

4.> Ângulo
5.# 0
Modifique o ângulo de 90 graus para 0.
6.> Feche a caixa de diálogo ***Texto***.
7.CA **Geral**
8.CO **Parcial** (painel **Selecionar elementos**)
9.+ Selecione o texto de conexão a ser girado.
10.+ Pressione a tecla CTRL e selecione outros textos caso queira girá-los ao mesmo tempo.
11.CA **Editar**
12.CO **Editar Texto** (painel **Texto**)
A janela ***Texto*** sera aberta.
13.> Ângulo
14.> 0
O ângulo será modificado nos textos selecionados.
15.> Feche a janela ***Texto***.

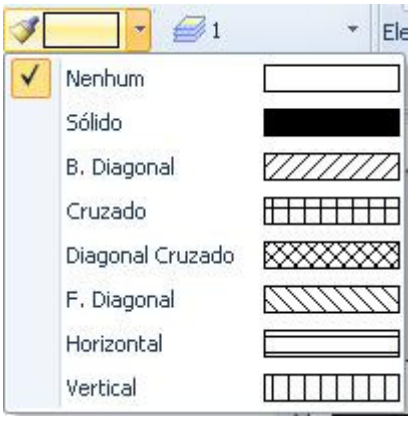
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

M.3. MANIPULANDO A PARTE GRÁFICA

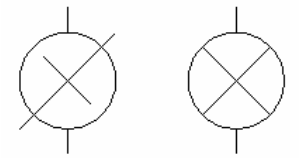
Exercício 12-11: Desenhar o símbolo.



Preencha o retângulo. Use o comando **Desenhar** □ **Elementos** □ **Preencher/Hachurar área**. Selecione o preenchimento adequado com o comando **Selecionar pincel** (painel **Desenhar** □ **Estilos**). Circunferências e elipses serão preenchidas automaticamente ao fazer o desenho, então o ajuste pode ser feito para "*Nenhum*".



Exercício 12-12: Desenhar o símbolo de uma lâmpada.

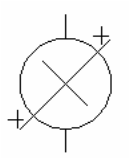


Não é possível desenhar retas que terminem no limite de uma circunferência sem mudar o valor do gri. Usando o comando **Editar** □ **Alterar elemento** □ **Aparar** é possível cortar linhas muito extensas. Com o comando **Editar** □ **Alterar elemento** □ **Extender** é possível prolongar uma linha até um limite determinado.

Para ajustar linhas que sejam muito longas:

1.CO **Aparar**
Ative a função a partir do painel **Editar** □ **Alterar elemento**.
2.+ Clique no limite (a circunferência).
3.+ Clique na parte a ser eliminada (o lado externo da reta).

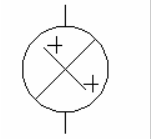
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



4.+ Clique no outro extremo da reta para eliminá-la.
Clique com o botão direito do mouse para cancelar a função.

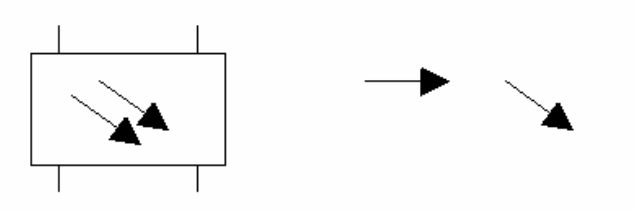
Para prolongar linhas que estejam curtas:

1.CO **Extender**
Ative o comando a partir do painel **Editar** □ **Alterar elemento**.
2.+ Clique no extremo da reta a ser prolongada:



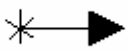
3.+ Clique no outro extremo da reta para fazer o prolongamento.
Clique com o botão direito do mouse para cancelar a função.

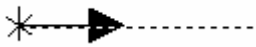
Exercício 12-13: Desenhar o detector de fumaça.



Desenhe as flechas com ângulo de zero graus. Desta forma elas podem ser desenhadas simetricamente. Use o comando **Editar** □**Ações** □ **Rotacionar** para girar os elementos.

Para girar as flechas:
1.+ Selecione a parte gráfica da flecha.
2.CO **Rotacionar**
Ative a função a partir do painel **Editar** □ **Ações**.

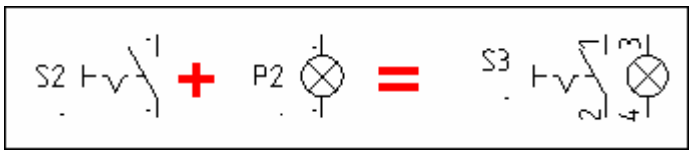
3.+ Defina o ponto que será o centro de rotação: 

4.+ Defina o eixo de rotação: 
5.+ Movimente o cursor. Os elementos serão rotacionados.
6.+ Posicione os elementos selecionados no ângulo desejado.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

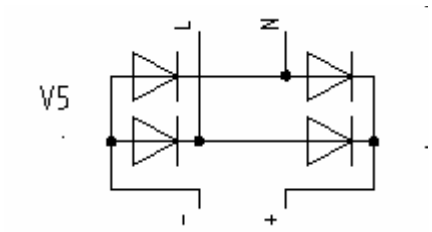
M.4. CRIANDO COMPONENTES A PARTIR DE OUTROS

Exercício 12-14: Criar um novo símbolo de lâmpada com sinalização usando o símbolo de botão NA com retardo ao abrir a partir da pasta Switchgear one pole e o símbolo de lâmpada a partir da pasta *Lamps*.



1. Insira os símbolos necessários a partir da biblioteca.
   Desagrupar o primeiro símbolo.
2.+ Selecione o símbolo
3. Clique com o botão direito do mouse e ative o comando Desagrupar a partir do menu suspenso.
   Desagrupe o segundo símbolo conforme descrito acima.
   Remova todos os elementos desnecessários ou desloque-os para um local livre da tela.
   Agrupe os elementos como **Componente**.
   Salve o símbolo na biblioteca.

Exercício 12-15: Desenhar um retificador. Use a parte gráfica de qualquer diodo presente na biblioteca como base.



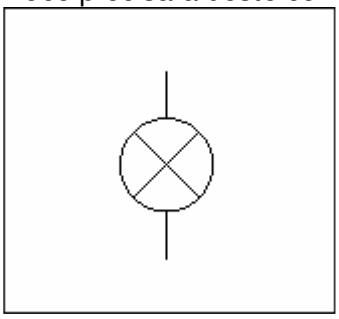
Atenção!!!

Desenhe os nós como circunferências preenchidas.

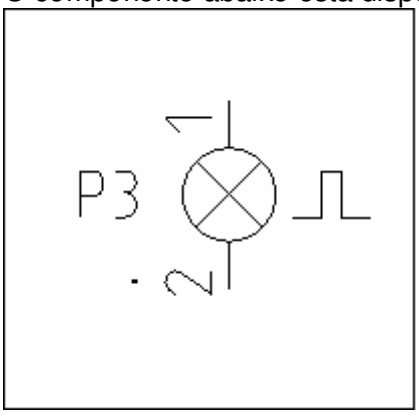
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercício 12-16: Se você precisar de um componente que pode ser facilmente criado após apagar elementos gráficos de um outro disponível, é possível proceder conforme abaixo:

Você precisará deste componente:



O componente abaixo está disponível:



- Insira o componente disponível.
- Clique com o botão direito do mouse para ativar o menu suspenso e acione **Selecionar parcial**. Após isso, selecione o elemento e apague-o. (É possível selecionar vários elementos mantendo a tecla CTRL pressionada.)

Se o símbolo for usado apenas no projeto atual não é necessário salvá-lo na biblioteca.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

M.5. INFORMAÇÕES SOBRE A CRIAÇÃO DE COMPONENTES ESPECIAIS

M.5.1. CONTATOS

Os contatos necessitam do texto de nome do componente, conexões e referência cruzada.
Se o contato possuir um campo de descrição, seu conteúdo virá da bobina.
No caso de contatos reversíveis, posicione os textos na seguinte ordem: comum, normalmente aberto e normalmente fechado.
O *SEE Electrical* também diferencia os contatos de força (normalmente aberto e normalmente fechado) e auxiliares (normalmente fechado, normalmente aberto e reversível).

M.5.2. BORNES

Os bornes necessitam de ao menos os seguintes textos: nome do componente, número do borne, índice do borne e ao menos um texto de conexão.

Para o número do borne você pode digitar qualquer número: ?, x ou X se o número tiver de ser incrementado. Desta forma você pode ter números de borne como L1, N ou PE.

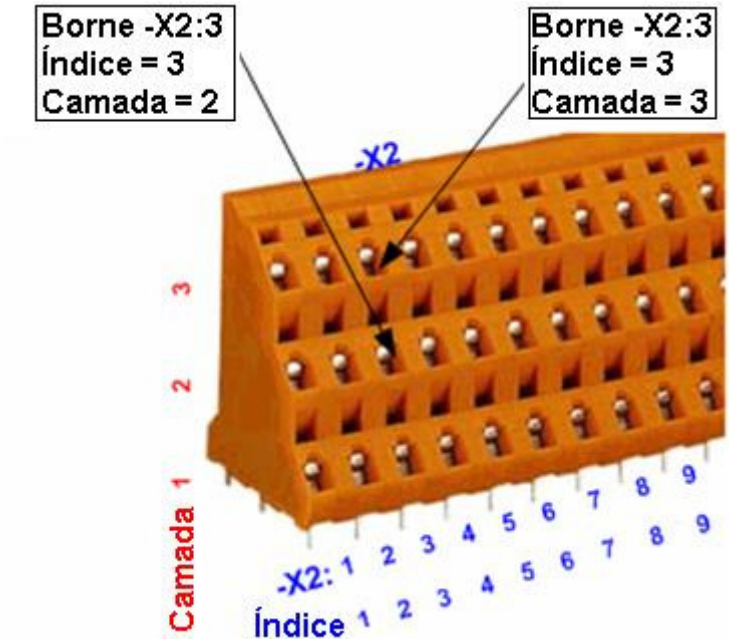
Se os textos forem posicionados automaticamente na criação do símbolo, o *SEE Electrical* posicionará as conexões automaticamente conforme descrito no parágrafo acima.

Se você quiser que o nome da régua e o número do borne apareçam em um texto (X2:17, por exemplo) você deve usar o atributo de texto **Nome do borne + número combinado** que pode ser encontrado na divisão *Outros*. Neste caso, posicione o texto com o referido atributo na parte gráfica. Após isso, agrupe o símbolo.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

M.5.3. BORNES MULTI CAMADAS (STANDARD)

Borrnes multi camadas representam vários bornes em um diagrama elétrico com o mesmo número e índice. A diferenciação entre eles será de acordo com a camada e o número em cada camada.



Todos os elementos de um borne multi-camadas possuem um mesmo tipo.
Na lista de componentes (nível *standard*), apenas um registro aparecerá. No modulo de *Layout* apenas um símbolo estará disponível.

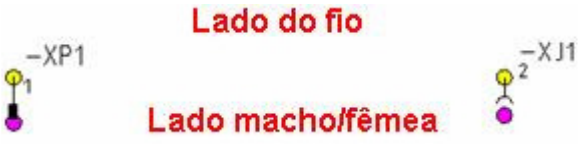
A diferença entre bornes multi camadas e “normais” é que o primeiro precisa de textos adicionais para identificar o número da camada e seu nome, assim como um separador entre o número do borne e o nome da camada.

M.5.4. TOMADAS

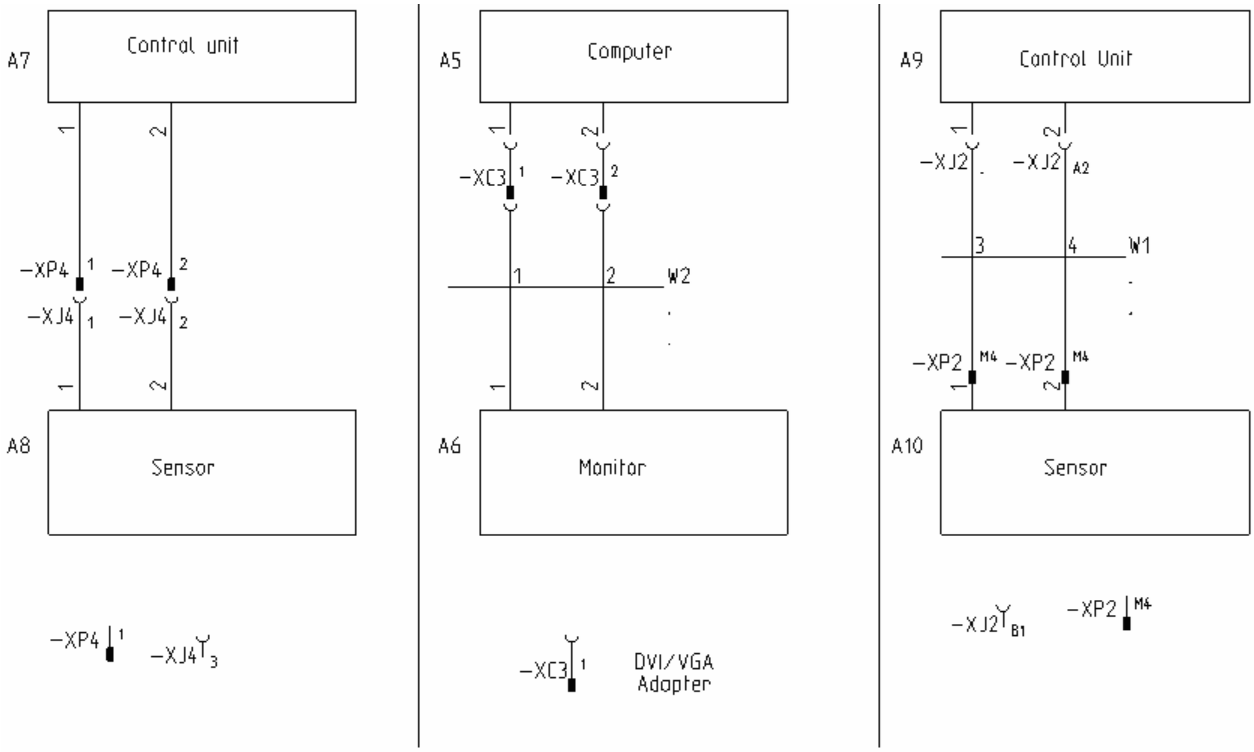
Tomadas necessitam de textos específicos. Na criação de uma tomada os textos de "Nome do componente", "Descrição", "Tipo", "Nome do terminal" e "Id do terminal (Ordenado)" são gerados automaticamente.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

O "Nome do terminal" contém o nome do terminal definido pelo fabricante.
A "Id do terminal (Ordenado)" determina a sequência na qual os terminais serão adicionados automaticamente ao esquema elétrico.
Se você quiser um nome diferente do sugerido automaticamente, modifique o “Nome do componente” na janela ***Propriedades do componente***.
Tomadas possuem a parte macho e fêmea. O plugue é conectado a um fio de um lado e à parte fêmea do outro. A parte fêmea pode ser parte de um componente ou de outra tomada.
A parte fêmea é conectada a um fio de um lado e a parte macho do outro.



No *SEE Electrical* o lado do fio deve ser representado pela primeira conexão e o lado do plugue pela segunda.
Nenhum fio deve ser conectado ao segundo ponto. Caso você desenhe uma conexão no lado do plugue, ele será usado apenas para endereçamento componente/tomada e inserir a informação na lista de tomadas, mas isso não sera mostrado na lista de fiação e tampouco será inserida numeração nele. Não é possível desenhar um cabo nesta situação.



Tenha o devido cuidado com os lados das conexões antes de salvar o símbolo na biblioteca (**Diagrama Elétrico Visualizar Pontos de conexão**). Se houver a necessidade de trocar as conexões, use o comando **Diagrama Elétrico Conexões Substituir**.

M.5.5. SÍMBOLOS DE REFERÊNCIA CRUZADA

A forma pela qual os potenciais são administrados no projeto definirá como usar símbolos de referência cruzada.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

✓ Símbolos de referência cruzada em projetos com potenciais

Símbolos de refência cruzada consistem da parte gráfica e textos de *Componente* e *Referência Folha/Coluna* (caixa de diálogo ***Texto*** □ *Atributos* □ *Componente*) e um símbolo de conexão. Se o símbolo de conexão for inserido manualmente, o campo ficará vazio. Se o símbolo for inserido automaticamente, o texto de conexão ficará visível. Neste caso, o texto de conexão deve ser apagado, de forma que apenas o símbolo de conexão permaneça. (Tecle F6 para ativar o comando **Selecionar parcial**, selecione o texto de conexão e apague-o).

Se vários símbolos de referência cruzada possuírem o mesmo nome é possível definir um índice para que exista uma sequência de vínculos entre eles. Deve haver sempre apenas dois símbolos de referência cruzada com o mesmo nome e índice pois esta é a única maneira para que exista o endereçamento correto.

O campo de texto de índice precisa ter o atributo "*Índice do componente*" (pode ser encontrado na divisão "Componente"). O texto deve ser posicionado manualmente antes que o símbolo seja agrupado.

Se você renomear um símbolo de referência cruzada que possua índice, apenas o nome do símbolo que possui o mesmo nome e índice será modificado.

✓ Símbolos de referência cruzada em projetos com potenciais representados por fios

Os símbolos de referência cruzada consistem da parte gráfica e textos de *Componente* e *Referência folha/coluna* (caixa de diálogo ***Texto*** □ *Atributos* □ *Componente*) e um texto de conexão.

Nestes símbolos há um texto de conexão que determina se um símbolo de referência cruzada deve ser reconhecido como origem (texto de conexão $1) ou destino (texto de conexão $0).

As referências cruzadas serão geradas se houver um par de símbolos de referência cruzada com o mesmo nome, onde um tenha o atributo de origem e o outro de destino. Um símbolo de origem sempre buscará o próximo destino com o mesmo nome – na próxima ou em qualquer outra página na sequência. (Isso permite que as referências cruzadas funcionem mesmo que uma página seja inserida ou removida). Os símbolos de destino fazem a busca em páginas anteriores a atual.

M.5.6. SÍMBOLOS DE TEXTO DE INFORMAÇÃO

Símbolos de texto de informação permitem administrar os destinos em listas como as de bornes, cabos, cabos/vias, fiação, multicondutores (apenas no nível *advanced*). A designação dos símbolos de texto de informação não é feita/modificada automaticamente.

Símbolos de texto de informação consistem da parte gráfica, texto de *Componente* e símbolo de conexão. O texto de conexão deve ser apagado deforma que apenas o símbolo de conexão permaneça (tecle F6 para ativar o comando **Selecionar parcial** e apague-o).

Nenhum caractere (ou espaço) deve ser inserido como nome do componente ao criar o símbolo para evitar a modificação automática do nome do componente. Ao criar o símbolo, não preencha o nome do componente, apenas clique em **OK**.

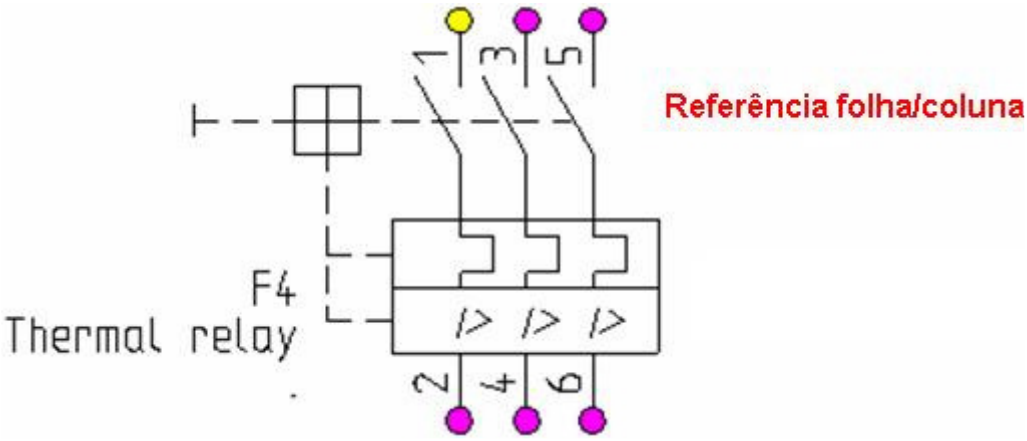
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

M.5.7. COMPONENTES COM CONTATOS AUXILIARES

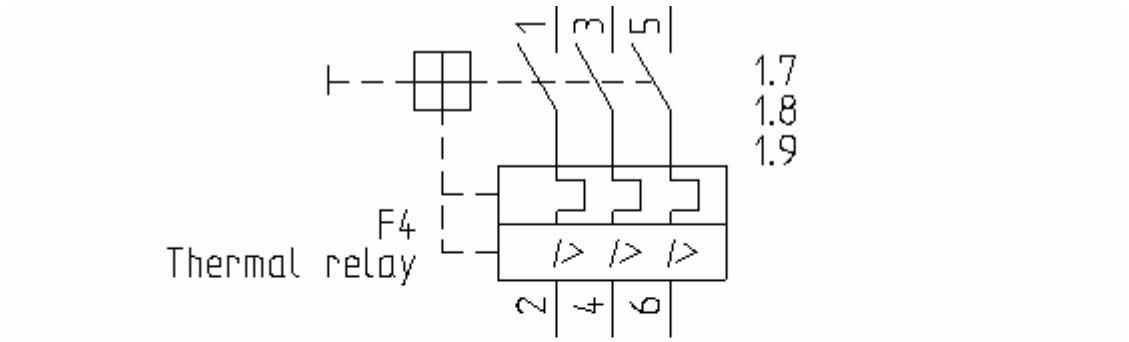
Há duas maneiras de criar componentes com contatos auxiliares:

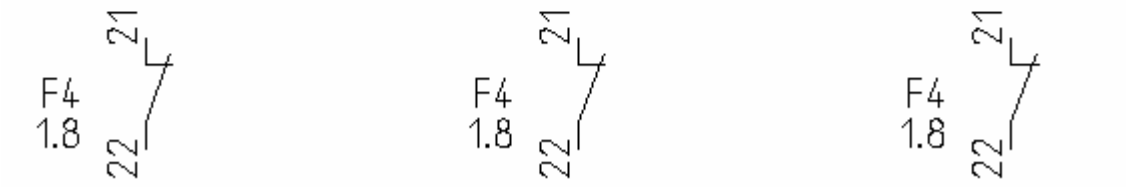
M.5.7.a. SÍMBOLOS COM TEXTO DE REFERÊNCIA CRUZADA

Um componente com contatos auxiliaries consiste da parte gráfica e textos de nome do componente, descrição 00, tipo, etc. Também serão necessaries os textos de "Referência cruzada" (é possível encontrá-los entre os atributos de texto da divisão "Componente"). Os elementos deverão ser agrupados como **Componente com contatos auxiliares** usando o comando **Editar** □ **Açõess** □ **Agrupar**.



Se você inserir no projeto contatos com o mesmo nome do componente principla, os textos de referêncua cruzada serão preenchidos. Segue exemplo:





O espaçamento de linhas varia conforme o posicionamento dos textos na criação do símbolo.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

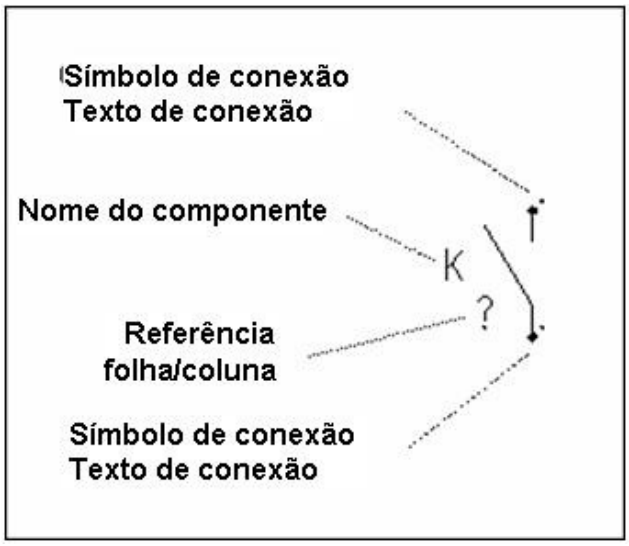
M.5.7.b. SÍMBOLOS COM CONTATO INTEGRADO

Um componente com contatos auxiliaries consiste da parte gráfica e os seguintes textos: nome do componente, descrição, tipo, textos de conexão, etc. Além disso, os contatos disponíveis no componente são necessários. Os itens devem ser agrupados como **Componente com dependência de contatos** usando o comando **Editar** □ **Ações** □ **Agrupar**.

M.5.7.c. EXEMPLOS DE COMPONENTES COM CONTATOS AUXILIARES

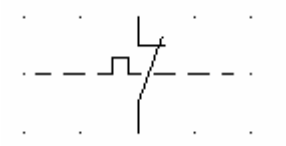
Contatos dos componente com contatos auxiliares
Se uma referência cruzada tiver de existir entre um componente com contatos auxiliaries e um contato você deve inserir os símbolos de contato no componente.
Primeiramente crie a parte gráfica.

Insira os textos. Preste atenção quanto aos atributos utilizados.



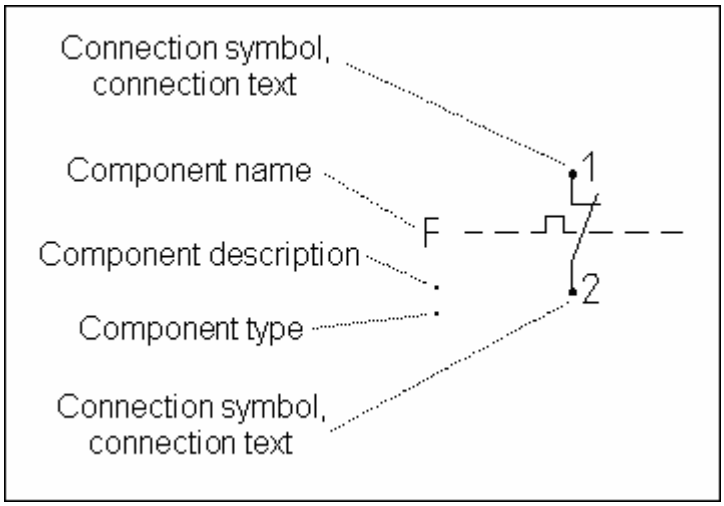
Desloque o texto de conexão, se necessário.
Selecione todos os elementos do símbolo e agrupe-o como Contato (utilize o tipo correto: normalmente aberto, normalmente fechado ou reversível).

Componente com contatos auxiliares
Desenhe a parte gráfica.

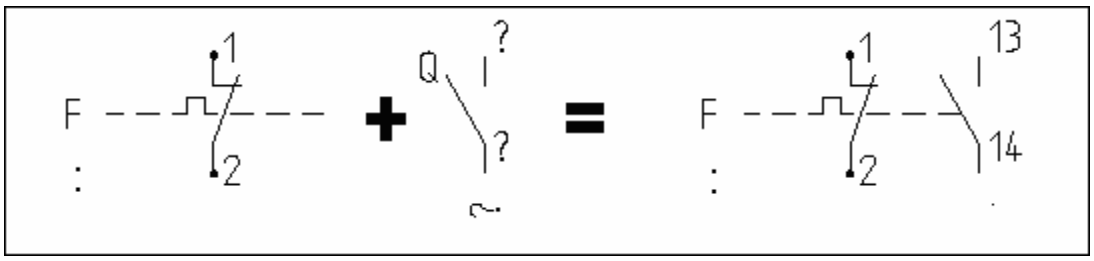


COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Create the texts. Pay attention that the texts receive the specified attributes.



Move the connection(s) text(s) if necessary.
Insert the required number of contacts. Insert the contact symbol without a text placeholder for the component name from the symbol database.



Select all the elements of the symbol and group them into a Component symbol with auxiliary contacts. *SEE Electrical* deletes the component names from the contact symbols.
Save the symbol in the symbol database.

M.5.8. PLC COMPONENTS

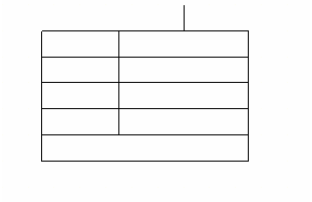
For PLC-components, *SEE Electrical* differentiates between PLC-racks and PLC signals (inputs or outputs).
When the symbols are created properly, the PLC-racks receive cross-references to the inputs and outputs, and the inputs and outputs receive a back reference to the Rack.
A single input/output always consists of the graphics and the following texts: component name, connection text(s), cross reference (optional, for the back reference to the Rack), Address, PLC comment (optional), description (optional). A PLC-signal can be created via the **Edit** □ **Actions** □ **Block** command.
A PLC-Rack consists of the component graphics, the texts for the component: component name, connection texts, comment and type and the number of PLC-signals (see above). You can integrate the single elements into a PLC-Rack using the **Edit** □ **Actions** □ **Block** command.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

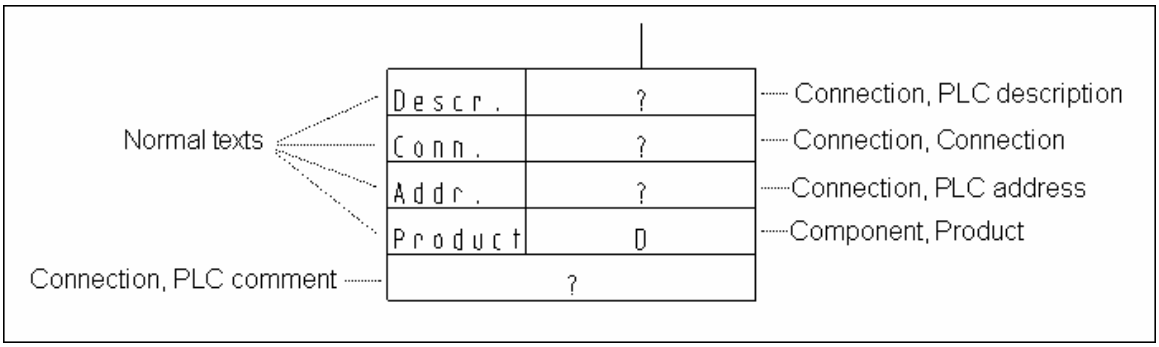
M.5.8.a. EXAMPLES OF PLC COMPONENTS

PLC-signals

- Draw the graphics.



- Create the texts. Make sure that the texts receive the specified attributes. All of the displayed texts including the texts for addresses and descriptions are necessary.



- Move the connection texts if necessary.
- Select all the parts of the symbol and create a PLC-signal symbol.
- Save the symbol in the symbol database.

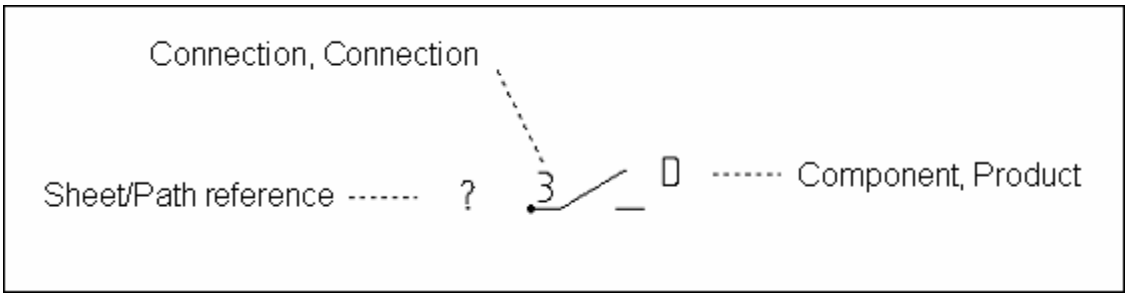
PLC-signals to be used in PLC-Rack

If a cross-reference must go out of a PLC-Rack to a PLC-signal, you should place symbols for PLC-signals in the Rack.

- Draw the graphics first.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

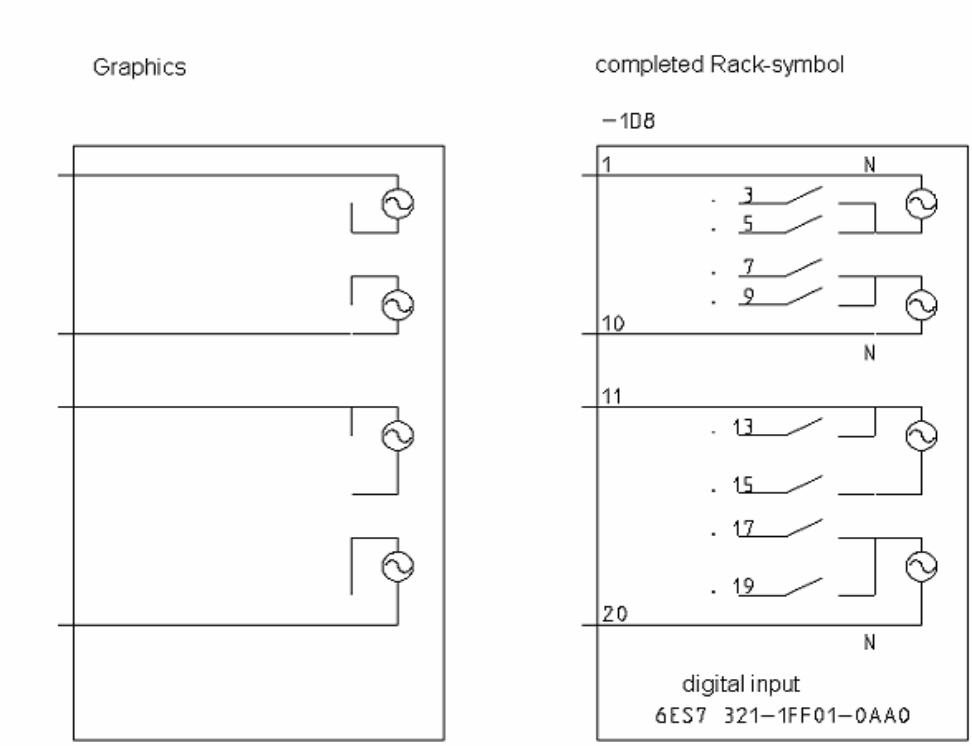
- Create the texts. Make sure that the texts receive the specified attributes.



- Move the connection text if necessary.
- Select all the parts of the symbol and create a PLC-signal symbol.

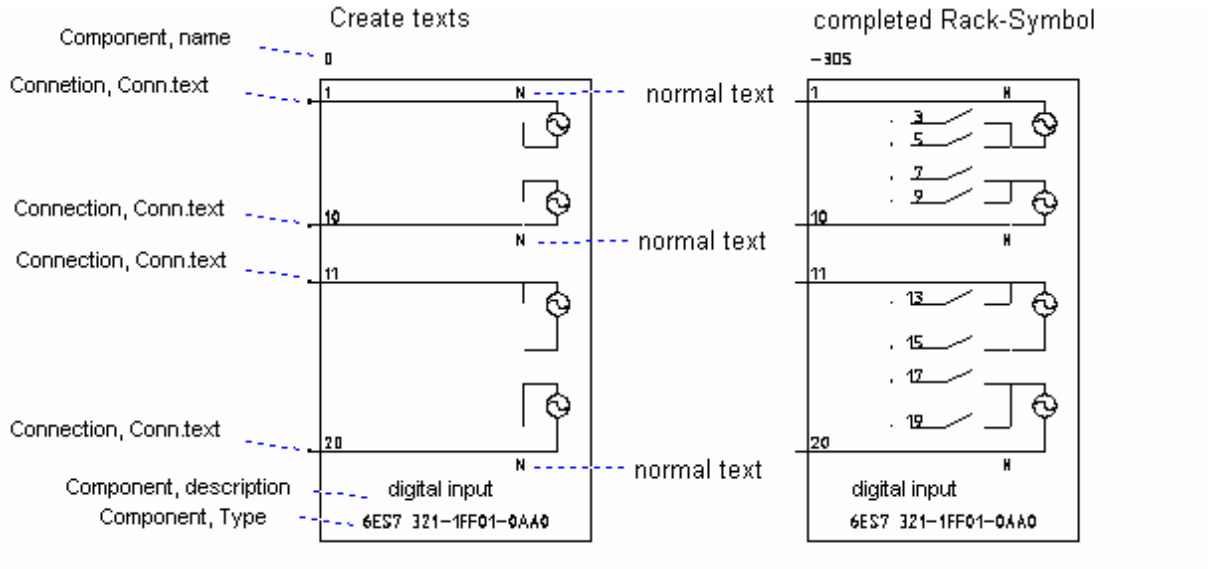
PLC-Rack

- Create the graphics first.

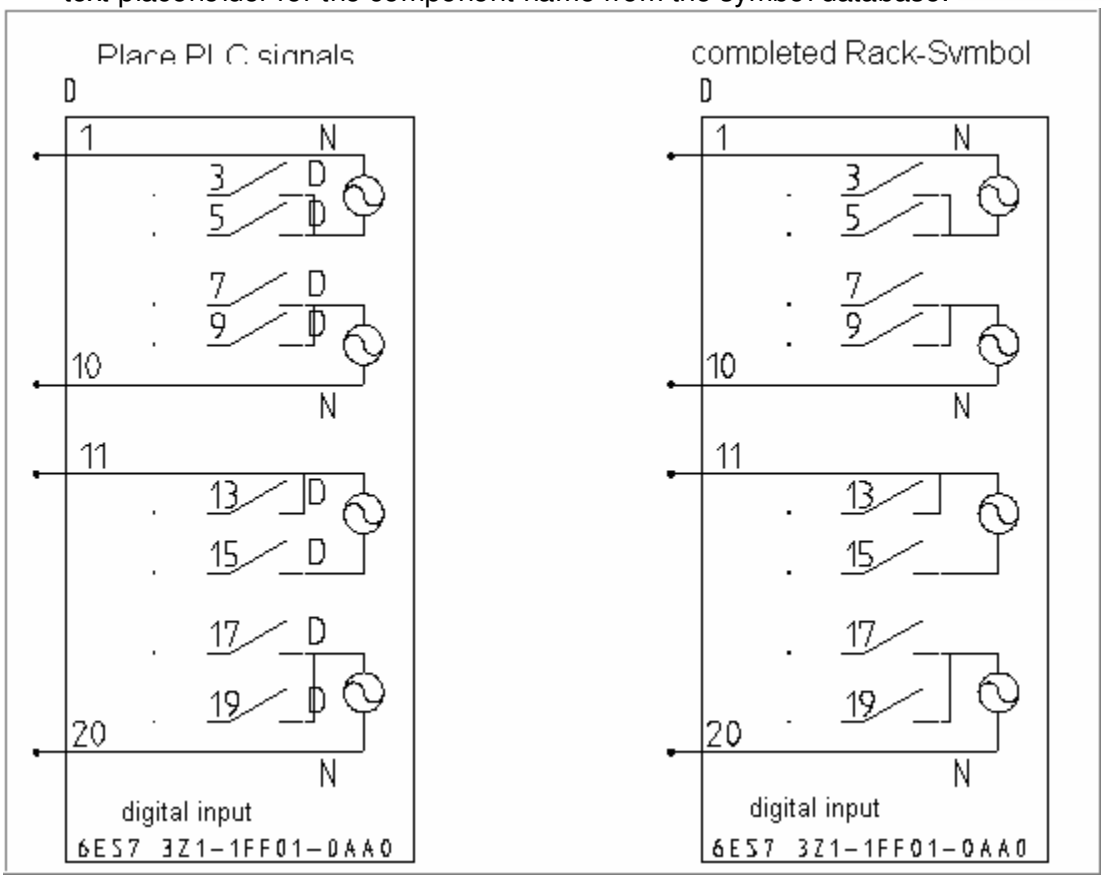


COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

- Create the texts. Make sure that the texts receive the specified attributes. At least one connection text is necessary. (If no connections are needed for the component, place the connection and define a space for the connection text).



- Move the connection text(s) if necessary.
- Place the required number of PLC-signal symbols. Insert the PLC-signal symbol without a text placeholder for the component name from the symbol database:



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

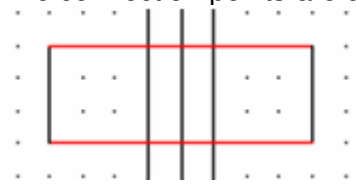
- Change the names of the PLC signals.
- Select all symbols and create a PLC-component symbol *SEE Electrical* deletes the component names from the PLC-signal symbols.
- Save the symbol in the symbol database.

M.5.9. SMART BLACK BOX SYMBOLS

The smart black box symbol creates connection points at the places where it connects with the wires.

1. To use smart black boxes draw a rectangle **Draw** □ **Elements** □ **Rectangle** around the wires.
2. Right-click it and select the **Block** pop-up command.
3. Choose the Smart black box symbol from the list.
4. Type in the code for the symbol.
5. Specify the lines that will automatically generate connection points.

The connection points are automatically created.



Hint:

*You can zoom in or out an intelligent black box symbol whose outline was generated as a rectangle, as follows: press the CTRL and SHIFT key on the keyboard at the same time and identify the rectangle while these keys are pressed. Only the rectangle is now selected and you can change its size when the default setting "**Trackers on the selected elements**" is active. Please make sure that you never change the size of an intelligent black box symbol when connections are already connected.*

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

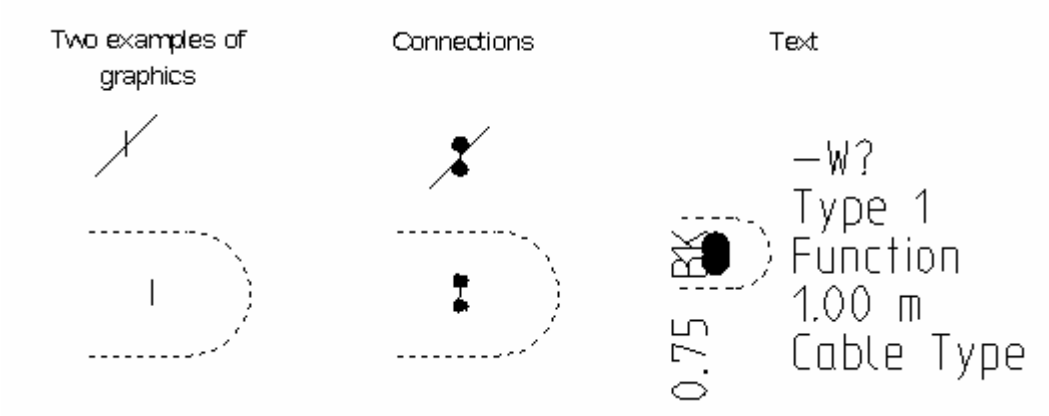
M.6. CABLE SYMBOLS

(Advanced)

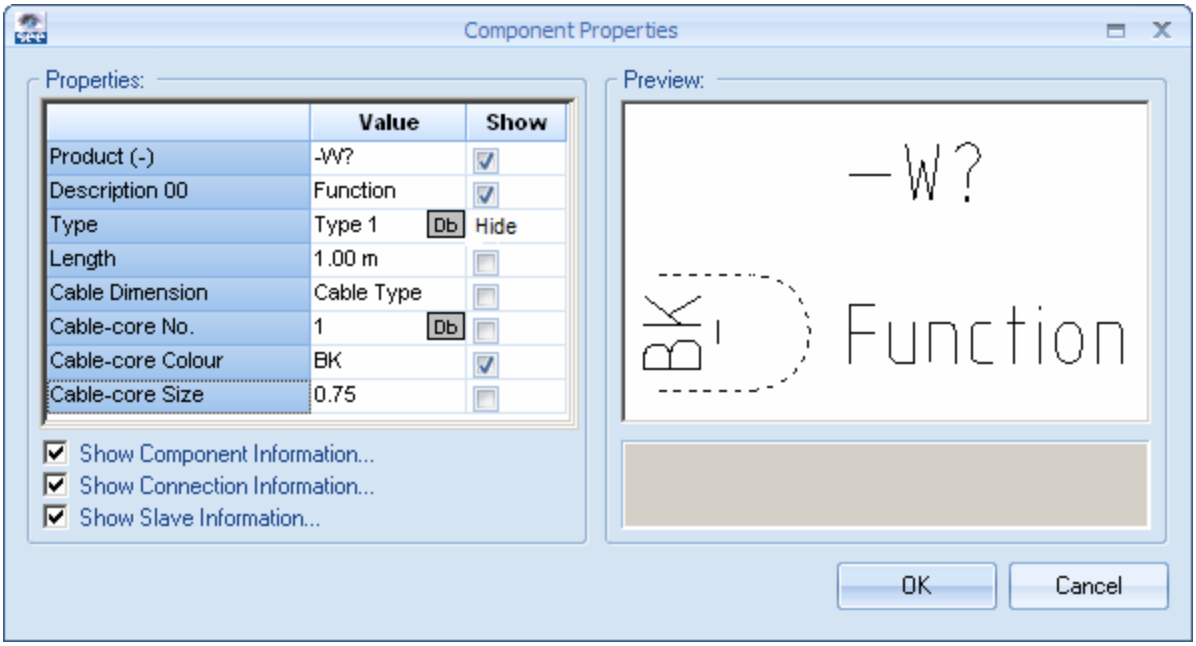
You can create user-defined cables and use them later when necessary.

M.6.1. CREATING USER-DEFINED CABLE SYMBOLS

Cable symbols consist of the graphic and the text placeholders for component name, function, type, length, 2 connection symbols (without text), cable core number, cable core colour and cable core section.
The symbol must be created with the correct angle.

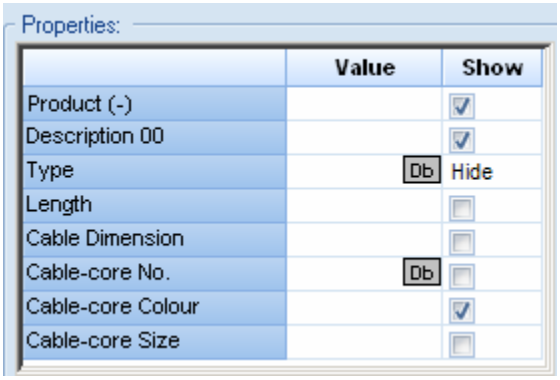


After grouping the elements into a cable symbol, you can define which texts will be visible via the check box in the "*Show*" column in the ***Component Properties*** window.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

- Delete all texts.



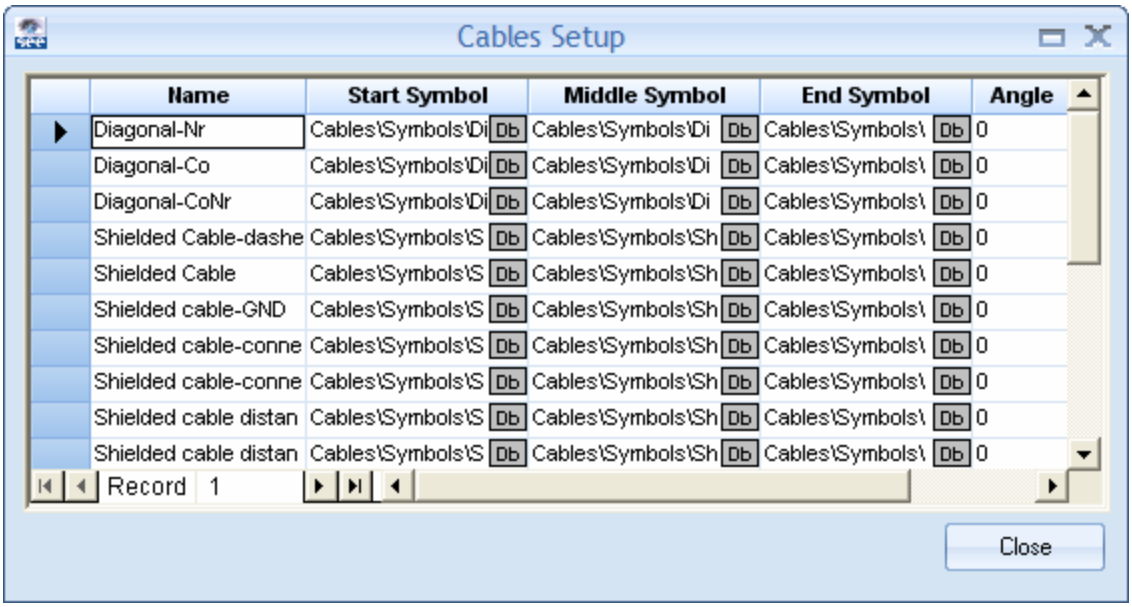
- Then move the cable symbol into the symbol library.

It is possible to insert the free texts 1 to 3 to the cable symbol. They also appear in the Cable list (database list and graphical list). These texts must be manually placed, if desired, before grouping the symbol.

M.6.2. CREATING USER-DEFINED CABLES

The functionality for creating your own cable definition and layout is accessible through the **Cables Setup…** button within the *Cables* tab of the ***Circuit Diagrams Properties*** dialogue. This dialogue is called by clicking the **Properties…** command from the pop-up menu that appears when you right-click the Circuit Diagrams module in the *Workspace Explorer*.

When you click the **Cables Setup…** button, the following window appears:



In the *Name* field, you have to specify a cable name by your choice.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

After you click the Db button within the "***Start Symbol***" field, the ***Symbol browser*** opens, allowing you to select the additional symbol used at the first cable core. Its description will be displayed in this field.

In the same way, you choose the additional symbol to be used at the middle core and the additional one for the last core of the cable. Their names (descriptions) are displayed, respectively, in the "*Middle Symbol*" and "*End Symbol*" fields. The definition is saved in *CABLESNEW.MDB*. The file must be located in your *Templates* folder. The definitions are available for all workspaces.

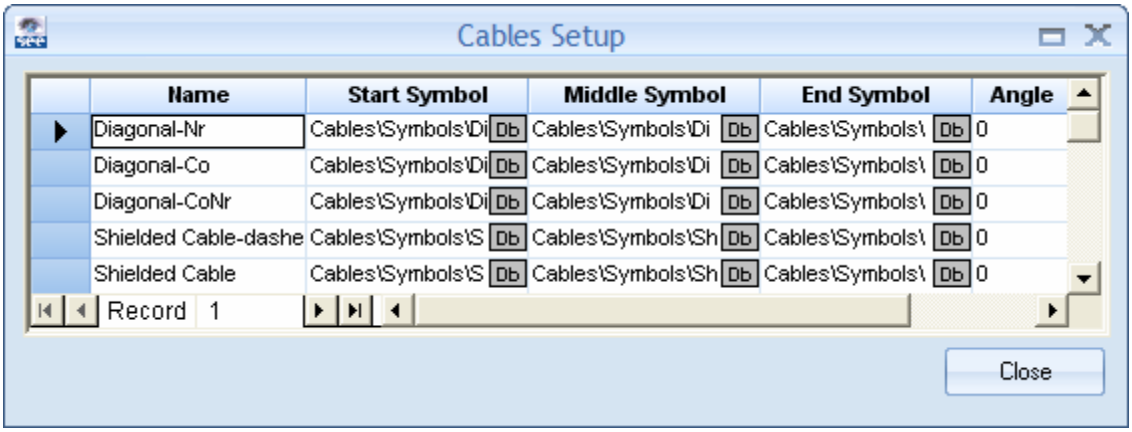
In the "***Angle***" field you have to specify the fixed rotation of your cable at the insertion. Because the symbols are used, the angle is fixed. If an angle of 0 degree is used, the cable cores are created from left to right. If an angle of 180 degree is used, the cores are created from right to left. The same is applied for 90 and 270 degrees.

Hints

1: For creating cable definitions, the various cable symbols must be present in the symbol database. Symbols must have all the necessary cable symbol properties assigned.
2: The symbols must be created at such angle as it is defined for use.
***3:** The defined cables are to be stored in the ...Template\CablesNew.mdb database. The cables from this database are available for all projects.*

When you want to use or add a user-defined cable, a dialogue listing all pre-defined cables appears.

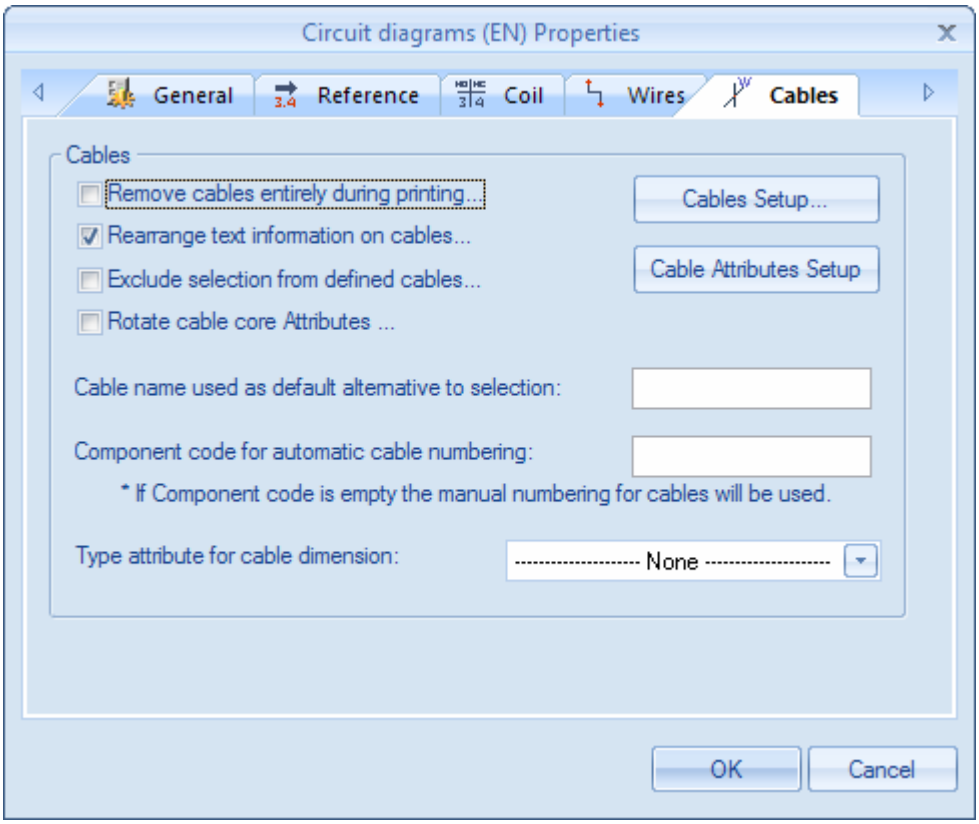
Example:



If no use-defined cable exists yet (possible list empty), no list will appear.
In case you choose a user-defined cable, it will be inserted. Otherwise (if you cancel the dialogue), *SEE Electrical* will add a basic cable.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

M.6.3. SETTINGS FOR CABLES



If you have created user-defined cables but you do not want to use them in a workspace, you can tick the "***Exclude selection from defined cables***" option.

If you have created user-defined cables and you want to use the same cable every time, you can type in the name of the desired cable in the "***Cable name used as default alternative for selection***" field:

Example:

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

M.7. INFORMATION ABOUT COMPONENT INSERTION

Components can be inserted from the symbol database into the drawing by left-clicking with the mouse to select the desired component and dragging it into the drawing (the left mouse button should not be pressed). The component is attached to the mouse and can be placed in the drawing.

M.7.1.a. RELAY COILS

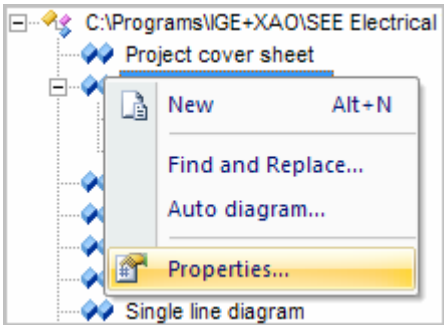
If a relay coil is placed, a contact cross appears beneath the component.
The contact cross can be moved independently, if needed.

M.7.1.b. TERMINALS

Terminals are named automatically using incrementing numbers, after they have been inserted. Terminals are sorted using an assigned index. The index is usually defined automatically. You can change the index as you insert the terminal or afterwards in the Component Properties dialogue. For example, the index allows inserting PE terminals in the appropriate place.

M.7.1.c. ASSIGNING A COMPONENT NAME

You can define the method of assigning component names in the Circuit Diagrams Properties dialogue.
Right-click the Circuit Diagrams module in the Workspace Explorer and select the Properties pop-up command.



Within the Component numbering area, select the desired component numbering.
For some symbols, such as contacts, it is possible to select the name of an available coil/relay or component with auxiliary contacts in the Product field. For terminals, you can select the name of an available terminal strip. For PLC inputs/outputs, you can choose the component name from the list of available PLC components.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

M.8. INFORMATION ABOUT SYMBOL DATABASES

M.8.1. CREATING A NEW DATABASE

If you want to create a new empty symbol database, right-click in the empty space of the Symbols area and execute the **New Symbol Database** function in the pop-up menu.

M.8.2. DIRECTORIES AND NAMES

SEE Electrical stores the symbol databases in the directory defined in the ***System settings*** dialogue, accessible via the **File □ System settings** button, which can be executed only if no project is open.
A database file has the *.SES* extension.
You can create symbols and save them in your own symbol databases. It is not allowed to perform changes in the supplied symbol databases marked with a icon.

M.8.3. WORKING WITH SYMBOL FOLDERS

When the symbol database or a symbol folder is selected, you can right–click and choose the **New Folder** pop-up command in order to create a new subgroup for components**.**
Type in the name of the **New Folder**.

The new symbol folder is arranged in alphabetical order in the tree of symbol folders.
If you wish to **Delete Folder** or **Copy Folder,** select the symbol folder, right click, and choose the appropriate command from the pop-up menu**.** To rename a symbol folder, choose the **Properties…** pop-up command and type in its new name.

A warning message appears before deleting, you must confirm this process.
The symbol folder has been deleted including all symbols within it.
The renamed symbol folder can be relocated to another place within the database because the symbol folders are arranged alphabetically.

After copying a symbol folder, it can be pasted into another place by using the **Paste Folder** command from the pop-up menu.

M.8.4. WORKING WITH SYMBOLS IN THE DATABASE

- Select a symbol and right-click to choose the respective pop-up command in order to delete, rename, and copy the symbol.

A renamed symbol can be relocated to another place within the database because the symbols are arranged alphabetically.
After copying a symbol, it can be pasted into another symbol folder by using the **Paste Symbol** command from the pop-up menu.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

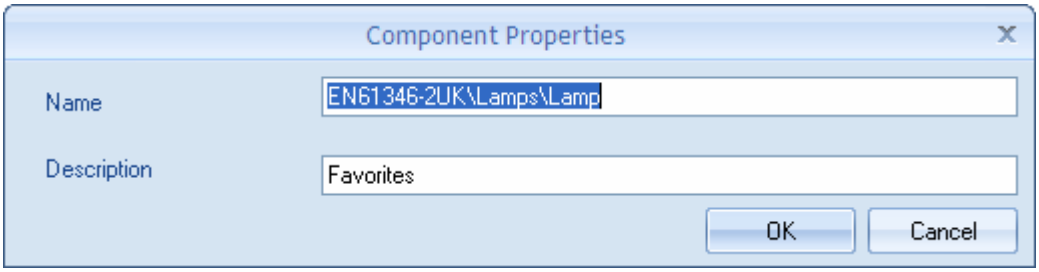
M.8.5. FAVORITES SYMBOL FOLDER

You can create your own, user-defined symbol folder, named "*Favorites*". By adding a symbol to this folder, you will create a shortcut to the referred symbol, whichever database it belongs to.
Right-click the desired symbol and execute the **Add to Favorites** pop-up command. The selected symbol is added to the *Favorites* symbol folder.

You can add any symbols, selected according to your needs, to the *Favorites* symbol folder.
Favorites are stored in the registry settings as a string list in the ***General\Favorites*** key.

The *Favorites* symbol folder is related to every module, e.g. *Circuit Diagram*, *Installation*, etc. It automatically expands, and clears, and collapses at filtering.

When you right click a symbol from this folder and execute the **Properties...** command, the favourite symbol properties – for the present case – will display the full link path.

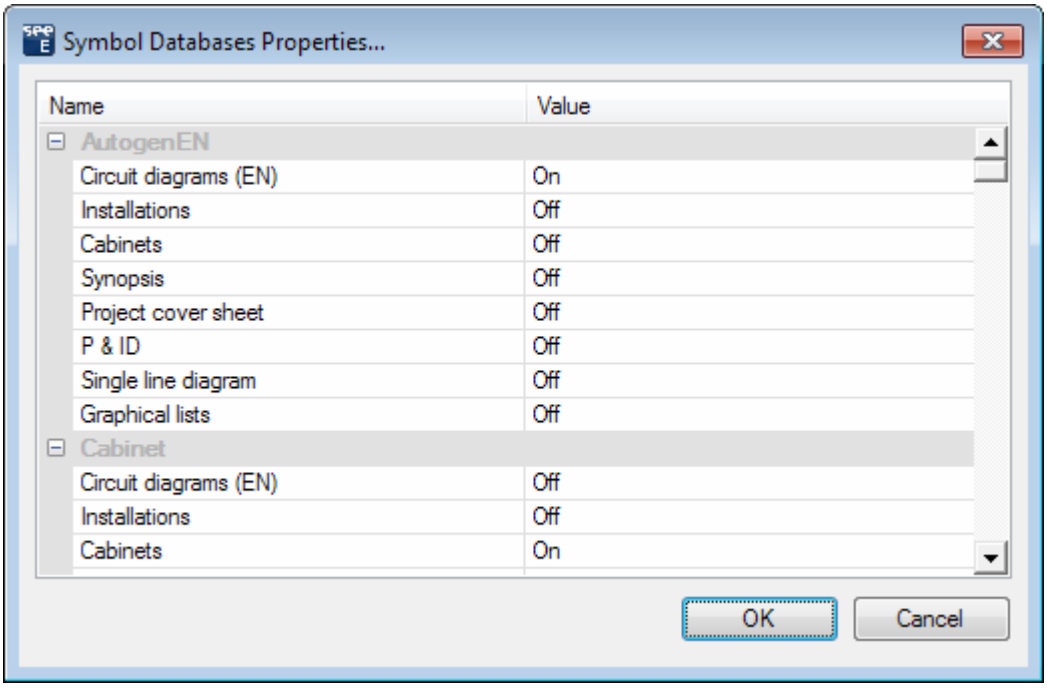


M.8.6. SYMBOL DATABASE CONNECTION TO MODULES

Since the number of the symbol databases within *SEE Electrical* is considerable, the symbol databases are allocated to the respective modules before they are displayed. So, a particular symbol database is connected to the module used.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

You can specify which databases to allocate. To do this, right-click within the **Symbol explorer** and select the **Properties...** pop-up command. In the window that appears, customize the visibility of the symbol databases (**ON** or **OFF**) and click **OK**.



This also means that the appearance of the ***Symbol Explorer*** (typically displayed in a more simplified and limited way) will vary depending on the module that you have activated, since all non-relevant symbol databases are left out.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

N GROUPS

Several components including wires and potentials (such as Reverse circuit breaker, interlock) can be saved in the symbol database.
In this way you can save time working with repeating circuit groups.
Before saving the group in the symbol database, you can choose its performance.

- ✓ ***Loose group***
  The single symbols, wires and potentials of the group are available for single manipulation, that is, they can be moved, copied or deleted.

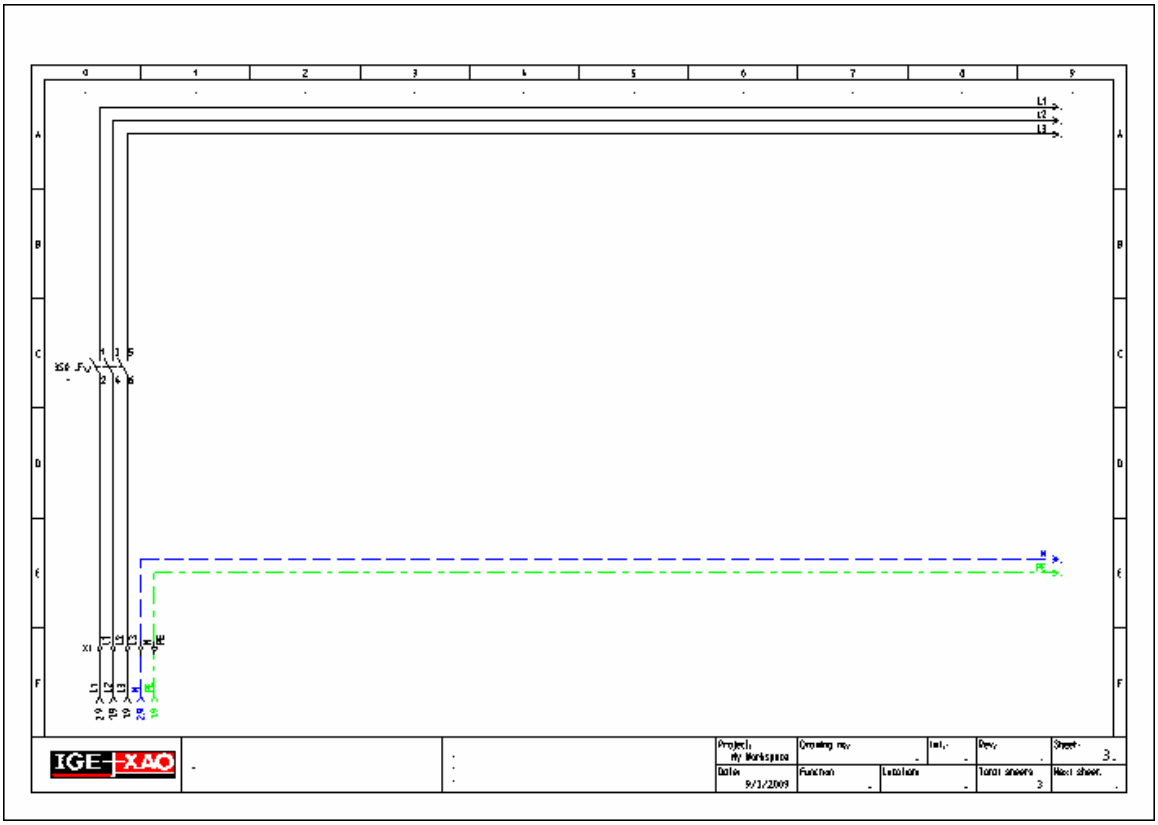
- ✓ ***Component group***
  The component group must remain grouped, i.e. it can be moved or deleted as a whole.
  After ungrouping, the single symbols, wires and potentials in the group can become accessible again and can be used individually, if needed.

In both cases you can edit the component names and the potential names.

N.1. CREATING A GROUP

Using the *Power supply* example, we will illustrate how the two kinds of groups are created.

Exercise 13-1: Draw the required potentials, insert the symbols for terminals and switchgear. Use the known commands.



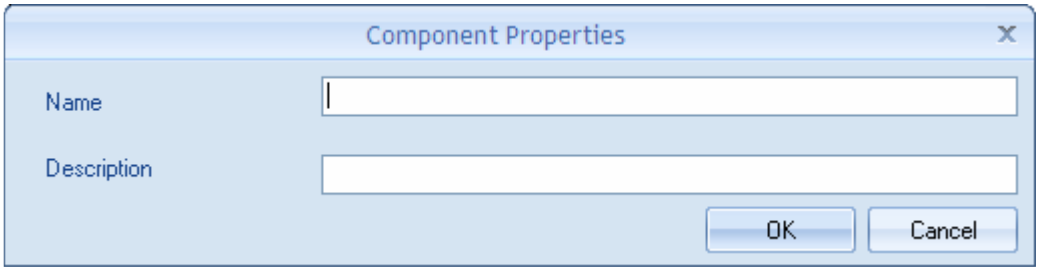
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercise 13-2: Save the group for further usage in the symbol database.

Before storing the group in the symbol database, decide what its performance will be:
Single symbols, wires and potentials can be manipulated single again, i.e. they can be moved, copied or deleted (**Loose group**).
The group must be manipulated as a whole (**Component group**), i.e. it can only be moved or deleted as a group. (If desired, it can be ungrouped so that single symbols, wires and potentials in the group can become accessible and can be used individually again.)
In both cases you can edit the component names and the potential names at any time.

Exercise 13-2a: First, save the supply as a loose group.

1. Activate the Symbols Explorer.
2. MySymbols
   Double-click the *MySymbols* database. You can save symbols only in this database or in a new one.
3. Right-click with the mouse.
4. CO **New Folder**
5. # Power supplies
   Type in the name of the new symbol folder.
6. CA **General**
7. CO **Normal** (**Select** panel)
8. \+ Select all the components of the power supply using a frame.
   Two ways exist for selecting part of the drawing:
   If you wish to process only elements that are located entirely within the area, move the cursor from left to right:
   The cursor graphic becomes:
   If you wish to process all objects that are even partly included in the area, move the cursor from the right to the left:
   The cursor graphic becomes:
   Select the components in the desired way.
9. \+ Drag the selected symbols into the newly created "*Power supplies*" folder. Hold down the left mouse button while dragging the symbol.



10. > Name
11. # Power supply 1
    Type the group's name.

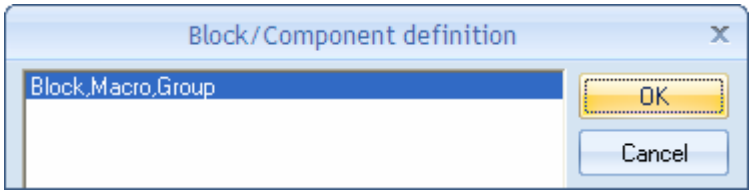
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

12.> **OK**
The group has been stored in the symbol database.
For example, if you delete page 3 in your project (using the **General** □ **Select** □ **All** and **Home** □ **Page** □ **Delete** commands), you can insert the group you have just stored. Afterwards, the switchgear can be selected and deleted separately. The group can be manipulated as a whole only if all the elements are selected within a frame beforehand.

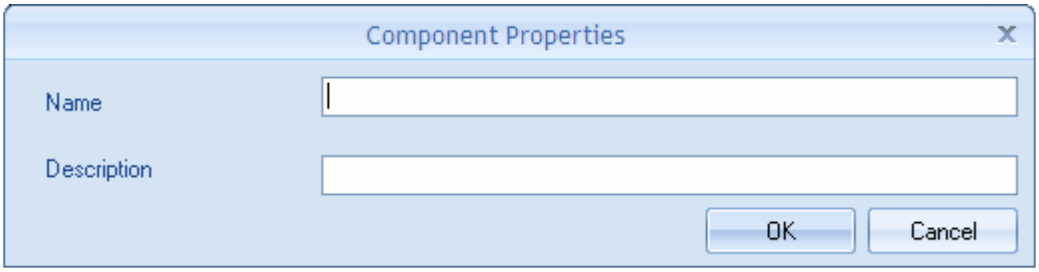
Exercise 13-2b: Now save the group as a component group.

Before saving, you must block the elements as a group.

1.CA **General**
2.CO **Normal** (**Select** panel)
3.+ Select the first point of the frame.
All of the elements have to be within the frame. You can use one of the two possible ways of selection, as described above.
4.+. Select the second point of the frame.
5. Right -click
6.CO. **Block**



7.> Block, Macro, Group
8.> **OK**
The parts of the power supply are grouped.
Save the group.
8. Activate the Symbols Explorer.
9. MySymbols
Double-click the **MySymbols** database to open it. You can store symbols only in this database or in a new one.
10.CA **General**
11.CO **Normal** (**Select** panel)
12.+ Click to select the newly created group. All the components belonging to the group are selected simultaneously.
13.+ Drag the symbol group into the newly created "Power supplies" symbol folder. Hold down the left mouse button while dragging the symbol.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

14.> Name
15.# Power supply 2
Type in the group's name.
16.> **OK**
The group has been stored in the symbol database.

Exercise 13-3: Ungroup the Power supply 2 group again.

Delete page 3 again (using the **General** □ **Select** □ **All** and **Home** □ **Page** □ **Delete** commands). Insert the **Power supply 2** group. Select the power supply. All of the elements can be selected only together. If you wish to select a single component, the group must be ungrouped first.

1.CA **General**
2.CO **Normal** (**Select** panel)
3.+ Select the group
4.CA. **Edit**
5.CO **Explode** (**Actions** panel)
The single elements are accessible again (symbols, potentials and wires).
Select the switchgear again. It is accessible now.

N.2. UNGROUPING SELECTED ELEMENTS

If a single symbol (such as a wire, potential) must be moved or deleted after inserting the group, the group has to be ungrouped. (This does not apply to loose groups.)

- Click to select the group.
- Execute the **Edit** □ **Actions** □ **Explode** command. The group has been ungrouped.

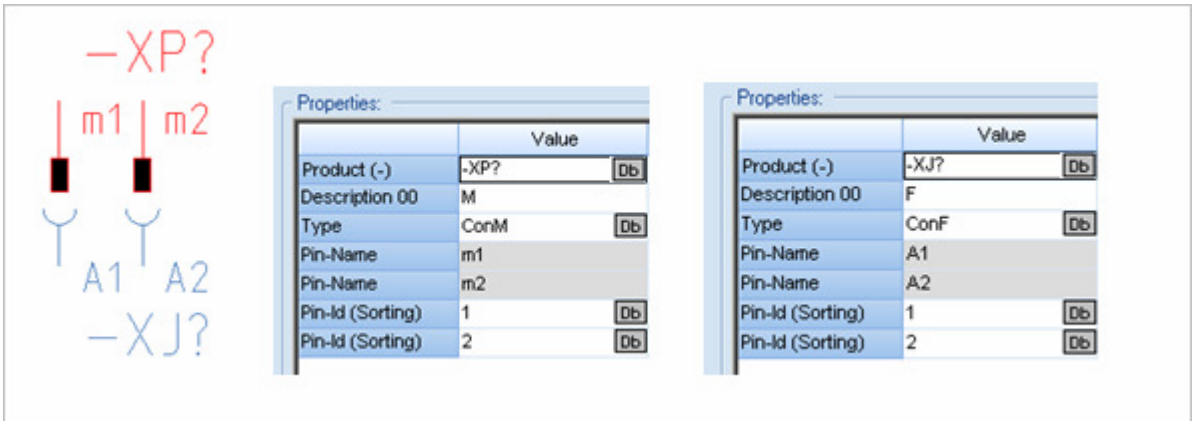
N.3. MOTOR TERMINALS AND SIMILAR COMPONENTS

If several terminals together are frequently needed, such as terminals for a motor, then those terminals must be defined as a group. For the second terminal and every next terminal, you can disable the view of the terminal strip using the **Edit Component** command.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

N.4. CONNECTORS: USING SEVERAL PLUGS AND JACKS TOGETHER

If you group several connector symbols as a macro/group, a ***Component Properties*** dialogue appears listing all pins of this group that belong to the same connector.
Example: Two macro groups have been defined and stored together in the library – one contains the male and the other the female pins.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

O BASIC OPERATIONS WITH TEMPLATES AND STANDARD SHEETS

There is a significant difference between templates for a workspace and templates for pages for a circuit diagram or for graphical lists in *SEE Electrical*.

Workspace templates
Workspace templates can be saved using the **File** □ **Save as** □ **Workspace template** command and they can later be chosen when you create a new workspace.

A workspace template consists of data about the component numbering, text size of cross-references, page template (norm sheet) for circuit diagrams, and templates for graphical lists.
A workspace template may also contain a completed standard circuit diagram to be inserted if necessary, saved under another name and then changed as desired.

Templates for single pages for circuit diagrams (Page templates)

Page templates can be saved using the **File** □ **Save as** □ **Page template** command and they can later be chosen using the **File** □ **Open** □ **Page template** command at any time.
A page template can be also assigned to the workspace template, then it will be used automatically for creating a new workspace.
In addition to the norm sheet, a page template contains the properties of the current page, such as page size, position of the first top potential, position of the first bottom potential, grid etc.

Templates for graphical lists

You can create your own templates for graphical lists. In addition to the graphics of the norm page, you can define which records from a database list must be recorded into the graphical list.
Before creating the graphical list, you can choose the template to be used. Select the respective graphical list in the ***Workspace Explorer***, right-click and execute the **Properties** pop-up command. Select the new template and close the dialogue box by clicking **OK**. Now the new template will be used if you create the graphical list.

The template chosen for each graphical list is saved in the workspace template. The templates for graphical lists have to be set before storing the workspace template. They are used by default for creating the relevant graphical list.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

O.1. CREATING A STANDARD SHEET

Standard sheets are created user-specific. This chapter will give you general information on how to create a standard sheet.

Exercise 14-1: Switch to an empty page of the workspace (there must be nothing in it except the standard sheet).

Delete the existing standard sheet using the **Edit** □ **Select** □ **All** and **Edit** □ **Actions** □ **Delete** commands. It is important that there is nothing on the current page.
You can disintegrate the existing standard sheet into its single parts later via the **Edit** □ **Actions** □ **Delete** command, right click and execute the **Explode** pop-up command, if you with to edit it and create a new standard sheet.

Exercise 14-2: Change the properties of your new standard sheet as desired. Define the number of columns in the drawing, too.

A standard sheet for circuit diagrams is usually drawn in the A3 or A4 format. For cabinet drawings you will need a standard sheet on a scale 1:1.

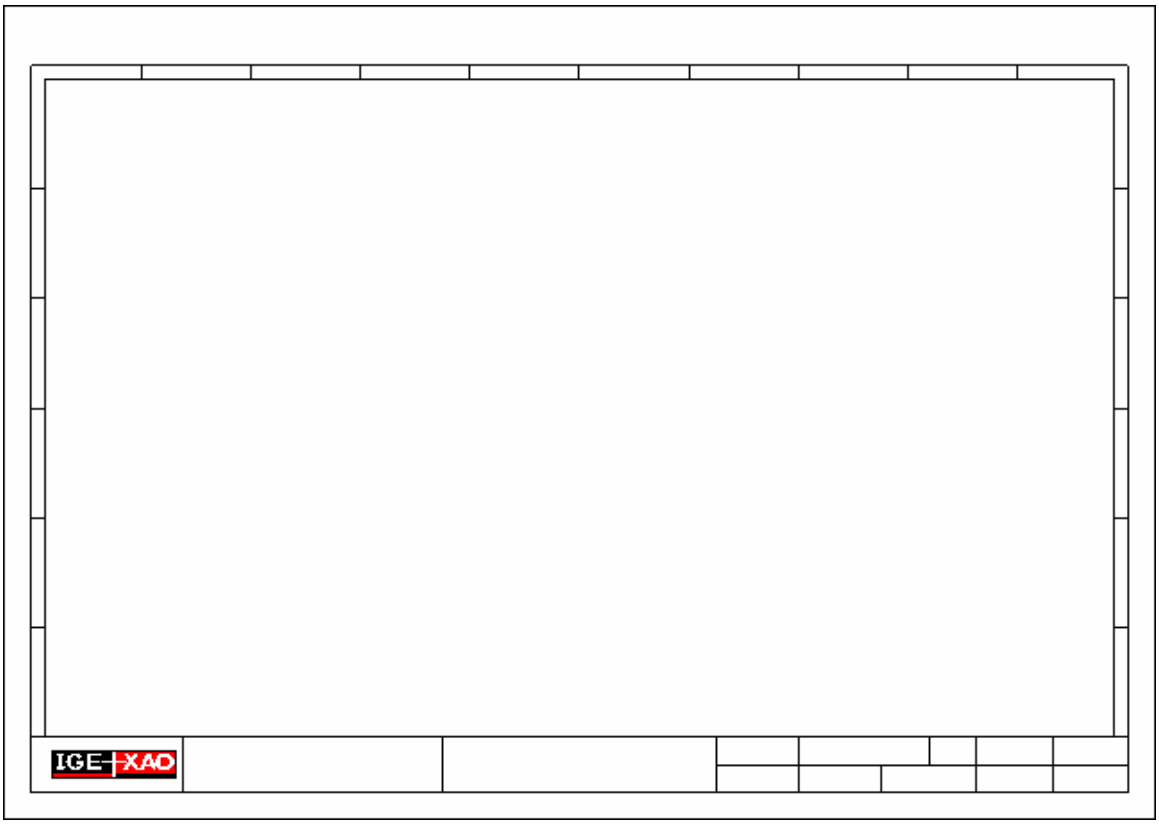
1. Select the page in the workspace tree that you are editing now.
2. Right click.
3.CO Select the **Properties** pop-up command.
The ***Properties*** window appears in the right pane of the main *SEE Electrical* window, displaying the properties of the current page.
4.> X-Extension of Page
Type in the new page size in the "***X-Extension of Page***" field.
5.# <new size>
6.> Y-Extension of drawing
Enter the new Y-extension of Page.
7.# <new size>
Look at the other settings in the dialogue box, position for the first top potential etc. Change other properties as desired. The position for the first top potential must be high enough, but within the section defined in the standard sheet. The definition of sections is described below.
8.> Press the Enter key to validate the new settings for each of the modified properties.
The properties are changed dynamically.

Exercise 14-3: Construct the graphics for your standard sheet (use the commands from the **Draw** category to draw a line, rectangle, etc.). Draw the geometry for the columns, too.

The complete standard sheet or your Logo can be imported using the **File** □ **Open** □ **CADdy Classic Drawing(s)** or **AutoCAD DXF/DWG Drawing** commands. You can find information about importing DXF/DWG/DXB drawings in the relevant chapter of this training manual.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

You can insert the company logo as a pixel image by using the **General** □ **Insert** □ **Picture** command. You can find information for inserting Bitmap Objects in the relevant chapter in this training manual.



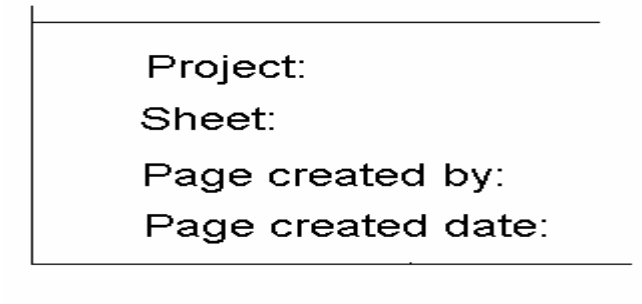
Exercise 14-4: Insert the texts you need into the norm sheet, such as project name, page name etc.
There is a difference between unchangeable texts and texts customizable to the current project or page. You can fill in the data of the current project as texts into the standard sheet, using text placeholders. For example, you can write a ? sign there.

First, insert the unchangeable texts.

1.CA **Draw**
2.CO **New Text** (**Elements** panel)
3.> Attribute
The unchangeable texts must have a "**Normal Text**" attribute.
4.> Text
5.# <Type the text>
6.+ Insert the text

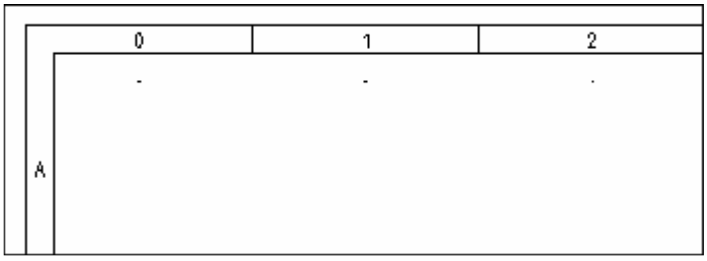
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Enter the fixed texts and place them into the desired positions.



To insert a text:
Type the text in the input field, for example: "Project:". Go out of the ***Text*** dialogue box with the cursor and place the text into the drawing. Go into the input field of the ***Text*** dialogue box again and type in the next text, etc.

Then insert the texts for the column. These texts have the "Normal text" attribute:



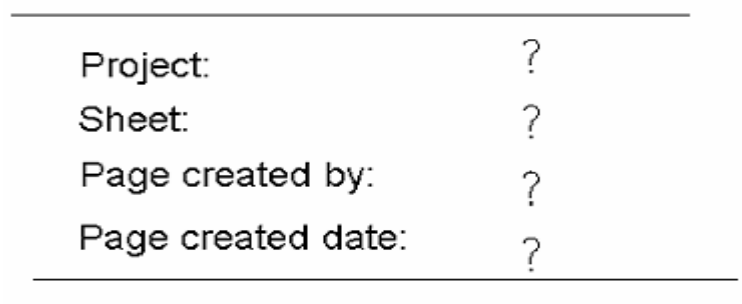
Next, insert the variable texts:
After you have inserted all of the fixed texts, place the changeable texts.

7.# ?
Type a question mark for the text placeholder.

8.> Attribute
Select the kind of the text placeholder.
The data from the ***Workspace Information*** window, which is available under the **Project** attributes node, belongs to the whole project and applies to all the pages.
The data from the ***Page Information*** window, which is available under the **Page** attributes, and it is specific for the current page.

9.> <Choose an attribute>

10.+ Insert the text
Choose the attribute for the changeable text, drag it into the drawing, and drop it on the desired location.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

To choose an attribute:

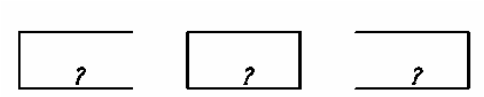
Choose the kind of text, for example **File-name** in the **Workspace** node of the attributes in the ***Text*** dialogue box. If you go out of the dialogue box, the question mark that you have entered in step 7 is attached to the cursor, and can be placed into the drawing. Go into the **Text** dialogue box again and select the next kind of text in the **Attribute** field, for example *Content / Page* or *Content / Page Created date***.** Place the question mark again. After you have inserted all texts, close the ***Text*** dialogue box.

Exercise 14-5: Select all of the elements that belong to the standard sheet. Group the selected elements to a standard sheet.

1CA **General**
2.CO **All** (**Select** panel)
3. Right-click with the mouse.
4.CO **Block**
Select this function from the pop-up menu.
5.> Page Template, Title block.
6.> **OK**

You have now created the first standard sheet. To use it for the next pages, create a page template.

Exercise 14-6: Define texts to use in headers for columns and rows in a symbol.



Example for the vertical symbols

Each symbol is made by lines.
Make sure all three symbols are of the same size, for example 10 mm wide and 5 high.
Each symbol has to contain the text to use (Text attribute "Column name marker" or "Row name marker".
1. Drag the upper left point to library for all the symbols.

Lines are extended / shortened if necessary. The Y-size of a column symbol is not changed (the same is for the X-size of a row symbol).

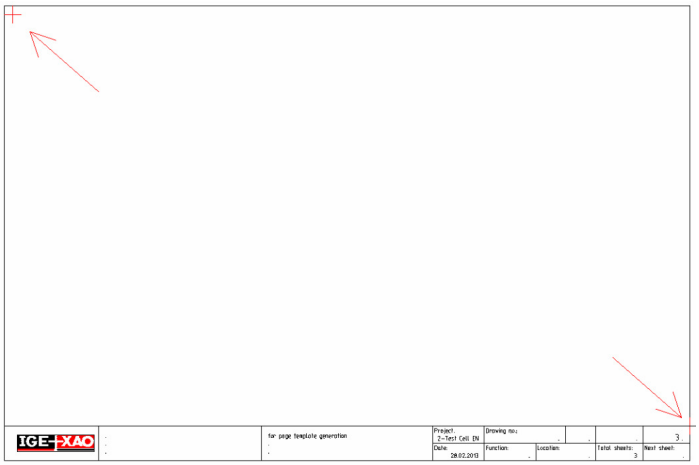
Exercise 14-7: You must now define the section to use on the page. Open the "***Page Properties***" again.

1.CA **Home**
2.CO **Properties**
3.> Click the the ... button in the ***"Page template sections"*** field.

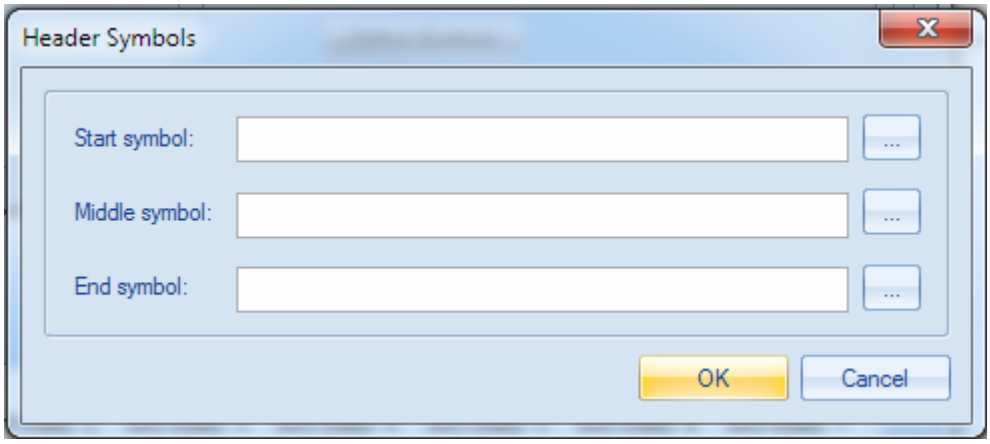
The ***Define Sections*** window appears. One section is always present by default.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

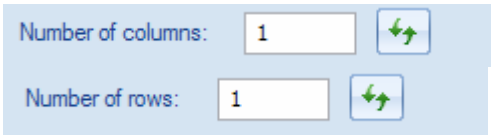
4.> Click the **Add** button to define another section.
5.> Insert a name for the section within the "***Name***" field.
6.> Click the button to define the area to use for the section on the page.
The area is defined by clicking to diagonal opposite points (like you do when drawing a rectangle).



After the area is defined, the X- and Y-coordinate of its start point and its width and height are displayed in the "***Size***" field of the window.
7.> Click the **Define Symbols** button if you want to use symbols.
The ***Header Symbols*** window appears.

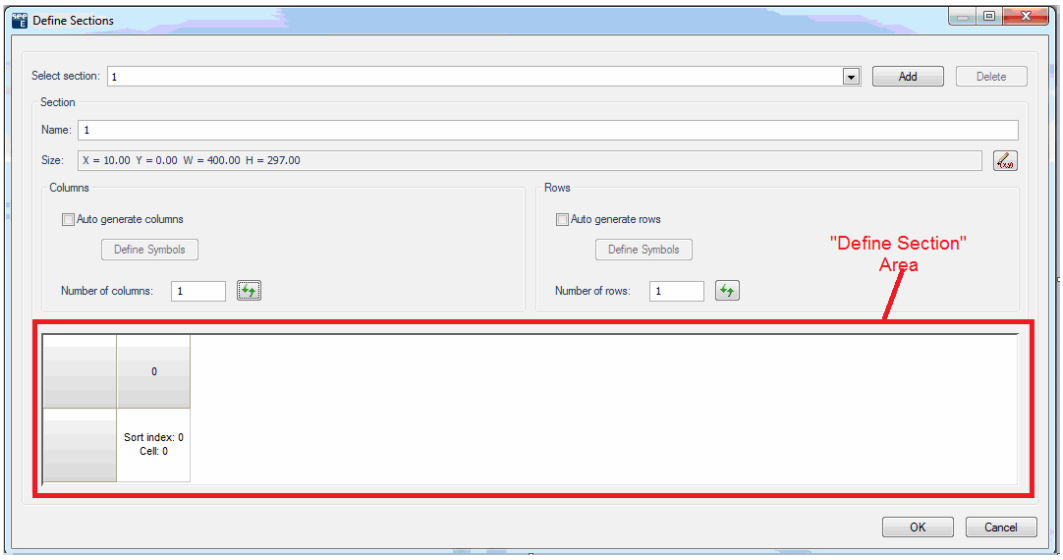


Three symbols are necessary, a start, middle and end symbol.
8.> Click the button to browse for symbols.
9.> Enter 1 as a value in the *"**Number of columns**"* and *"**Number of Rows**"* fields.

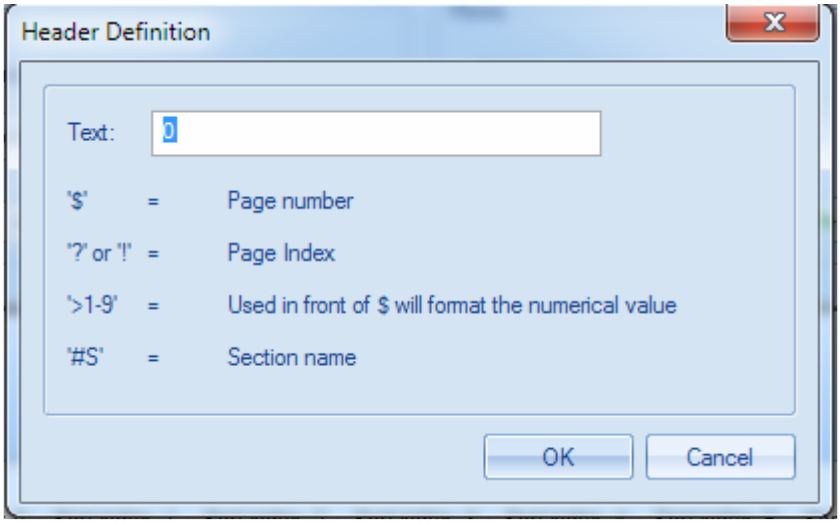


10.> Click the **Refresh** button to apply the changes to both
In ***Define Section*** area only one cell is visible now:

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

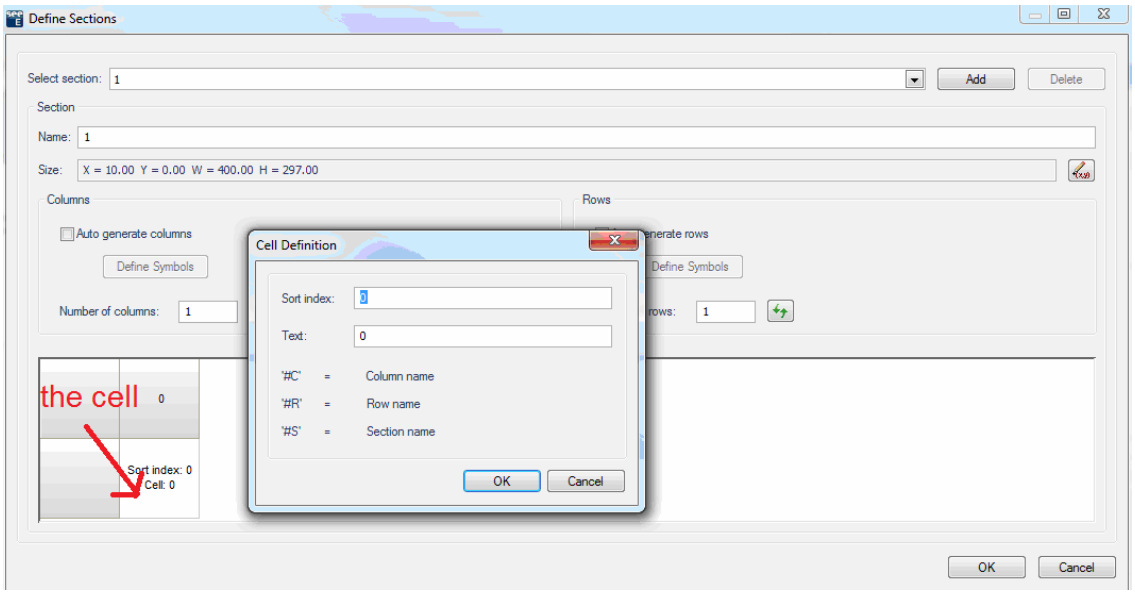


11.> Double-click the column header of the existing column to define the rules for the column.


Header Definition
Text: 0
'$' = Page number
'?' or '!' = Page Index
'>1-9' = Used in front of $ will format the numerical value
'#S' = Section name
OK
Cancel

12.> Enter 0 (zero) as the column name.
13.> Click **OK**.
14.> Define the rules for the row in the same way. (For example, add an *A* for the row name).
15.> Double-click the cell to define its rules.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



16.> Insert 0 (zero) for the Sort index to sort from top to down and from left to right.

17.> The content of the text field is used if the cell (column/row) information will be used in the names of the components or cross references. If you want to use thus value, you can define the cell information. If you do not want to use the cell information in the component names or cross references, leave the default value in the "***Text***" field.

18. Now define the number of columns and rows you want to have:

Example 1: only columns

Nothing is defined in the header of the rows. You will define only the "***Number of columns***" and will leave the "***Number of rows***" value to 1.

Use the **Refresh** button to generate the number columns.

Example 2: columns and rows

For example 5 rows and 10 columns will be defined: it is important to create the rows first and then the columns if you want to sort your objects in the database lists from the top left corner of a sheet down first and then to the right like it is done in the default templates delivered.

- Define the rows.
- Use the **Refresh** button to apply the value.
- Define the columns.
- Use the **Refresh** button to apply the value.

The columns and rows are automatically made. The rules for name of first column and row are used for the automatically made ones.

19. **Please control the names of the columns and rows and the order and names of the cells!**

If you made a mistake, you can change the rule for each of the columns or rows, but it is easier to start with step 7 from this description again.)

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

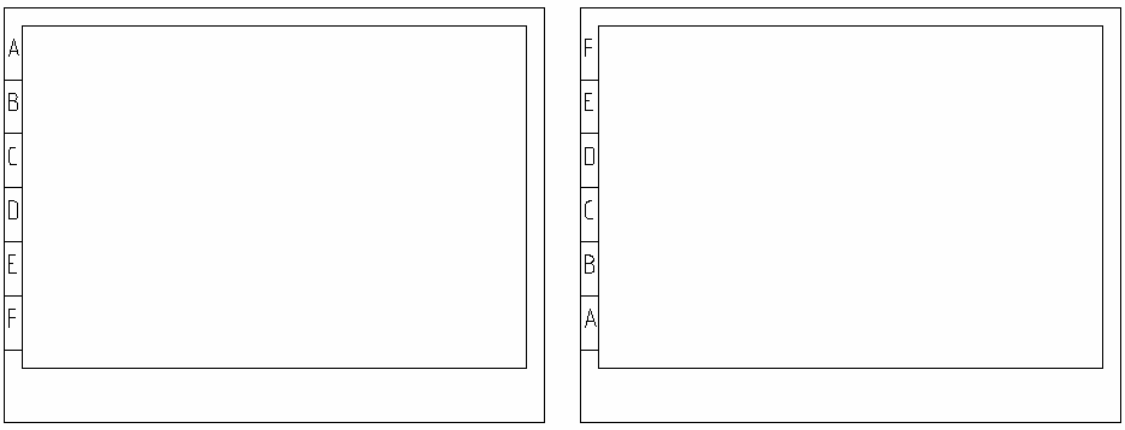
Example for columns only:

	0	1	2	3	4	5	6	7	8	9
	Sort index: 0 Cell: #R	Sort index: 1 Cell: #R	Sort index: 2 Cell: #R	Sort index: 3 Cell: #R	Sort index: 4 Cell: #R	Sort index: 5 Cell: #R	Sort index: 6 Cell: #R	Sort index: 7 Cell: #R	Sort index: 8 Cell: #R	Sort index: 9 Cell: #R

Example for columns and rows (= page coordinates)

	0	1	2	3	4	5	6	7	8	9
A	Sort index: 0 Cell: #C-#R	Sort index: 6 Cell: #C-#R	Sort index: 12 Cell: #C-#R	Sort index: 18 Cell: #C-#R	Sort index: 24 Cell: #C-#R	Sort index: 30 Cell: #C-#R	Sort index: 36 Cell: #C-#R	Sort index: 42 Cell: #C-#R	Sort index: 48 Cell: #C-#R	Sort index: 54 Cell: #C-#R
B	Sort index: 1 Cell: #C-#R	Sort index: 7 Cell: #C-#R	Sort index: 13 Cell: #C-#R	Sort index: 19 Cell: #C-#R	Sort index: 25 Cell: #C-#R	Sort index: 31 Cell: #C-#R	Sort index: 37 Cell: #C-#R	Sort index: 43 Cell: #C-#R	Sort index: 49 Cell: #C-#R	Sort index: 55 Cell: #C-#R
C	Sort index: 2 Cell: #C-#R	Sort index: 8 Cell: #C-#R	Sort index: 14 Cell: #C-#R	Sort index: 20 Cell: #C-#R	Sort index: 26 Cell: #C-#R	Sort index: 32 Cell: #C-#R	Sort index: 38 Cell: #C-#R	Sort index: 44 Cell: #C-#R	Sort index: 50 Cell: #C-#R	Sort index: 56 Cell: #C-#R
D	Sort index: 3 Cell: #C-#R	Sort index: 9 Cell: #C-#R	Sort index: 15 Cell: #C-#R	Sort index: 21 Cell: #C-#R	Sort index: 27 Cell: #C-#R	Sort index: 33 Cell: #C-#R	Sort index: 39 Cell: #C-#R	Sort index: 45 Cell: #C-#R	Sort index: 51 Cell: #C-#R	Sort index: 57 Cell: #C-#R
E	Sort index: 4 Cell: #C-#R	Sort index: 10 Cell: #C-#R	Sort index: 16 Cell: #C-#R	Sort index: 22 Cell: #C-#R	Sort index: 28 Cell: #C-#R	Sort index: 34 Cell: #C-#R	Sort index: 40 Cell: #C-#R	Sort index: 46 Cell: #C-#R	Sort index: 52 Cell: #C-#R	Sort index: 58 Cell: #C-#R
F	Sort index: 5 Cell: #C-#R	Sort index: 11 Cell: #C-#R	Sort index: 17 Cell: #C-#R	Sort index: 23 Cell: #C-#R	Sort index: 29 Cell: #C-#R	Sort index: 35 Cell: #C-#R	Sort index: 41 Cell: #C-#R	Sort index: 47 Cell: #C-#R	Sort index: 53 Cell: #C-#R	Sort index: 59 Cell: #C-#R

20. If you want to change the name of a column or row now, just double click its header and make your changes. Like this it for example is possible to have characters from F to A in the rows instead of A to F.



In this case it could be necessary to change the sort order too.

21. Click **OK** to close the window..

22. Now setup/control the other page properties as usual.

23. Keep in mind the positions for the first top and bottom potentials in relation to the section borders. The position for the first top potential is set relatively to the top border of section, the position for first bottom potential is set relatively to bottom border of section. The margins for the left and right potentials are also relative to the section borders.

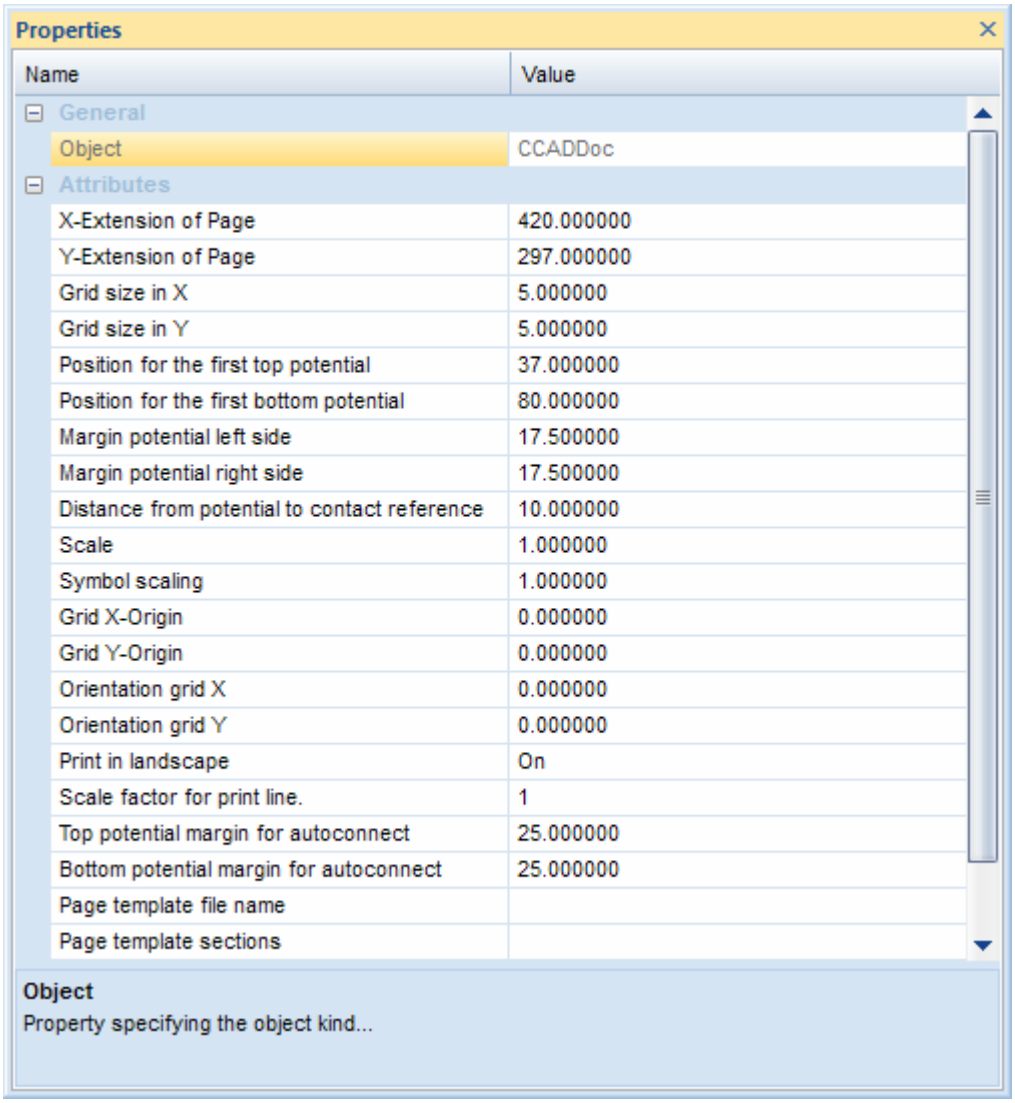
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

O.2. CREATING A PAGE TEMPLATE

You can save the standard sheet created in the previous section as a page template. Before you save the page template, look at the current properties, because a page template consists of the standard sheet and the appropriate properties.

Exercise 14-8: Verify and change, in necessary, the properties for the page template.

1. In the Workspace tree, select the page that you are currently editing.
2. Right-click with the mouse.
3.CO Execute the **Properties** pop-up command.
The ***Properties*** window appears in the right pane of the main *SEE Electrical* window, displaying the properties of the current page.
Look at the properties that you can set. Change some properties, as desired.



4.> Press the Enter key to validate the new settings for each of the modified properties.
The properties are changed dynamically.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercise 14-9: Save the page template.

1. In the Workspace tree, select the page that you are currently editing.
2.CA **File**
3.CO **Save as**
4.CO **Page template**
5.# <name>
6.> **OK**.
The page template has been saved. You can load it at any time or assign it as a page template to a project template, as described in the next chapter.

Exercise 14-10: Open the page template on another page.

1. <page>
In the Workspace tree, select the page that you are currently editing (page 1 or 2).
2.CA **File**
3.CO **Open**
4.CO **Page template**
5.> <Your template>
6.> No
The new page template has been loaded, the project data and page data have been inserted into the new standard sheet, and the existing circuit parts on the page have been kept.

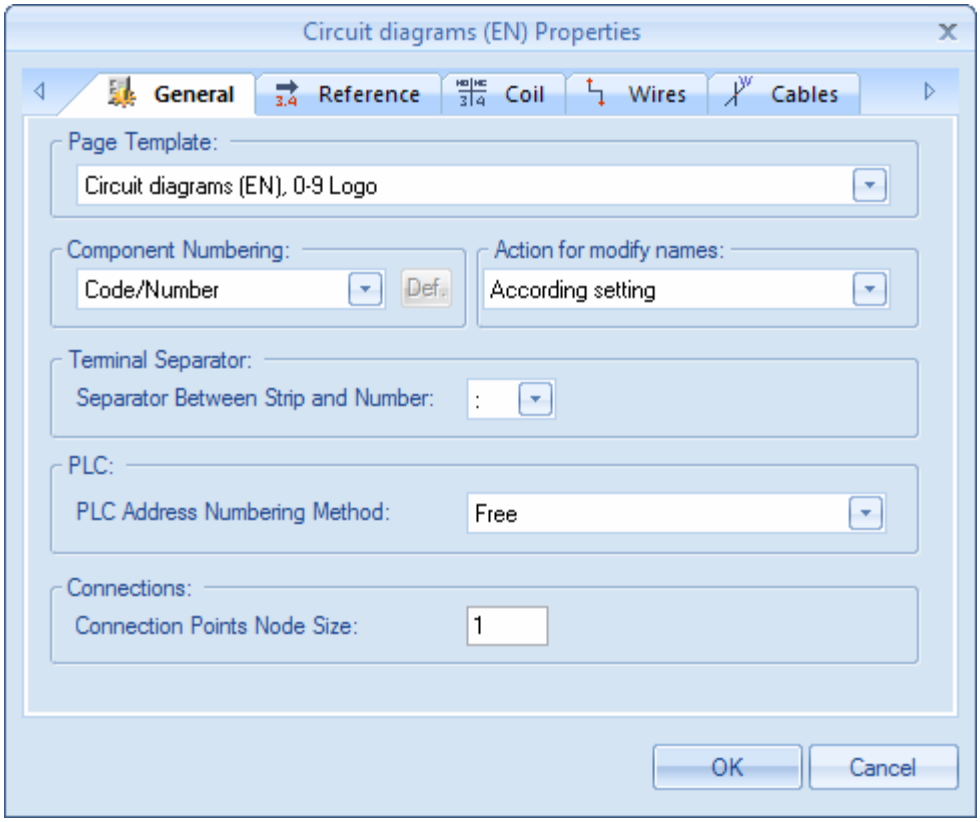
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

O.3. CREATING A WORKSPACE TEMPLATE

Exercise 14-11: Open an existing workspace template, change it and save it with another name.

1.CA **File**
2.CO **Open**
3.CO **Workspace**
4.> Click the ▼ in the "Look in" field of the dialogue box

You can search projects in another directory. By default projects are saved in the...*Projects* directory, templates - in the...*Templates* directory.
5.> Select the...\Templates directory under *SEE Electrical*.
6.> *SEE Electrical*
Select a workspace template. Its properties appear in the right pane of the ***Open Workspace*** dialogue.
7.> Open
The workspace template is opened.
8. Select **Circuit diagrams** in the Workspace tree.
9.CA **Home**
10.CO **Module** (**Properties** panel)



11.> Page Template
12. Select your page template.
Verify all of the properties in this window and the properties of the circuit diagrams.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Select a method for naming the components in the workspace from the **Component Numbering** pull-down list within the ***Circuit Diagrams Properties*** dialogue box.

13.> **OK**

Exercise 14-12: Save the new workspace template.

1.CA **File**
2.CO **Save as**
3.CO **Workspace template**
4.# <name of the template>
5.> Save

If you create a new workspace by using this template, the newly created page template applies to all new circuit diagram pages of the workspace. If you need to use different templates on different pages, it is possible to change the page template via the **File** □ **Open** □ **Page template** command, as described above.

Exercise 14-13: Close the workspace template to prevent undesired changes.

1. Click on the workspace name in the Workspace tree.
2. Right-click with the mouse.
3.CO **Close Workspace**

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

O.4. QUICKREFERENCE TEMPLATES AND STANDARD SHEETS

Workspace Template (-> **File** □ **Open** □ **Workspace**, Select the ...\Templates directory, Select template <Name>.*SEP*)

Workspace Properties (-> <Name>.*SEP* in the ***Workspace Explorer***, pop-up menu)
Function / Location settings
Revision

Circuit Diagram Properties (-> Circuit diagrams in the ***Workspace Explorer***, pop-up menu)
Component Numbering, etc.
Select a page template.

Page Template (-> <Name>.TDW)

Create Standard sheet
Draw graphics
Block: Page Template, Title block

Page Properties (-> Page in the Workspace Explorer, pop-up menu)
Number of columns in page
Position of Potentials
Grid size

Save Page template (-> **File** □ **Save as** □ **Page Template**)

Cabinets Properties (->Cabinets in the Workspace Explorer, pop-up menu)
Select page template, etc.

Page Template (-> <Name>.TDW)
See above

Graphical Lists – List of documents - Properties (->List of Documents in the Workspace Explorer, pop-up menu)
Select page template, etc.

Page Template (-> <Name>.TDW)
See above

Graphical Lists - List of products – Properties (->List of Products in the Workspace Explorer, pop-up menu)
etc.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

P TEMPLATES AND STANDARD SHEETS FOR GRAPHICAL LISTS

P.1. CREATING TEMPLATES FOR GRAPHICAL LISTS

Exercise 15-12: Create templates for graphical lists.

1. Open the Graphical Lists area.
2. Select the graphical list you wish to create a template for.
3. Right-click with the mouse.
4.M Select the **Load Page Template** command from the pop-up menu.
The newly created template can be loaded and changed. After you delete this template, it is possible to create a new one. (The template opens in its own window, i.e. it does not exist as a page in the Workspace Explorer).
It was described in the **Creating a Standard Sheet** chapter how to delete an available standard sheet using the **Edit** □ **Select** □ **All** and **Edit** □ **Actions** □ **Delete** commands.
Now, you must change the available standard sheet.
5CA **General**
6.CO **All** (**Select** panel)
7. Right-click
8.M **Explode** from the pop-up menu.
After ungrouping the standard sheet, all the elements become accessible again.
9. You can add, erase and move lines, if needed.
10. You can also edit, erase and add texts.
There is a difference between unchangeable texts and texts that have to be entered with data from the database lists.
All of the texts in a template for graphical lists have the "Normal Text" attribute.
Texts to be filled in with data from the database lists, must be presented always in the format #<number>, such as #120010 for page number. You can find a list of the available numbers for text placeholders at the end of this section or in the "*List Construction Set.SES*" symbol library.
11. A special text placeholder "#Lines 30 7.5" must be presented in the template in order to state the number of lines available in the template (for example 30) and the line-distance (for example 7.5).
Enter the text exactly in the following format:
#Lines <number of lines> <line-distance>
The point is required as a decimal delimiter in the line-distance.
After you have created the template, group it again.
11CA **General**
12.CO **All** (**Select** panel)
14. Right click
15.M Select **Block** from the pop-up menu.
16.> Page Template, Title block
17.> **OK**

Exercise 15-13: Save the template.

1. In the Workspace Explorer, select the current page under Graphical Lists.
2.CA **File**
3.CO **Save as**

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

4.CO **Page Template**
5.# <name>
6.> **OK**

Exercise 15-14: Select a template for a graphical list as follows:

You can select the template for the graphical list in the ***Properties*** dialogue box.

1. In the Workspace Explorer, select the graphical list, which you would like to select a new template for.
2. Right click
3.CO **Properties**
4.> Select your new template.
Click the icon to select the desired template. A pop-up list appears, including all of the templates for graphical lists.
Select the template.
5.> **OK**
The setting for the template has been saved within the workspace.
If you create a new workspace template, you can select new templates for graphical lists, too.

Examples for text placeholders from database lists:
You will find detailed information about all the lists in **The Graphical Lists** chapter of the software's Help.

List of products

Number	Description
120010	Page number
120020	Page Index
160020	Cell
140020	Function (=)
140050	Location (+)
160010	Product
160030	Description
160040	Type1
160041	Type2
160042	Type3
160043	Type4
160044	Type5
160045	Type6
160046	Type7
160047	Type8
160048	Type9
160049	Type10
165150	Free text 01
165151	Free text 02
12000170	Manufacturer

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

P.2. FORMATTING TEMPLATES FOR GRAPHICAL LISTS

The format of graphical lists (except for Terminal matrix, Terminal plan, Terminal row picture, Cable Terminal row plan, connector matrix, connector plan and Assembly list) can be customized by using: sorting, filtering, defining first page, page break after changing the defined values.
The control is realized via the page template using some keywords defined with the "#" sign:

```
#PageBegin 100
#PageBreak 140020, 140050
#Orderby 140020, 140050, 160010
```

```
#Lines 30 7.5
#WHERE [140020]="=A1" AND [120010]>10
```

Function (=)	Location (+)	Product (-)
#140020	#140050	#160010

#PageBegin <page number>	The graphical list will start from the defined page number. Example (for List of Products): #PageBegin 100 The first graphical List of Products always begins automatically from page number 100. #PageBegin ? Each time you generate the List of Products, a question about the start page number will appear. Using #PageBegin [<List ID>] it is possible to assign consecutive page numbers to all graphical lists. Example: First, the Document List is created. The page numbers of the Product List will be followed on seamlessly to the last page of the Document List. So the placeholder # Page Begin [3011] is required in the Standard sheet symbol of the page template for the Product list. For the IDs of the various lists, see the chapter "**Generate all desired graphical lists in one step** " below. It is possible to define one page number and different indexes for pages that are generated for one kind of a graphical list. In this case, the page number will not be changed. The definition has to be made in the following way: #PageBegin <page number> <separator> <page index> The page index has values in the range from A to Z. The numbering is executed in the same way as the numbering for the Excel columns. For example: #PageBegin 10 / A -> the result is pages 10.A, 10.B, 10.C.....

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

	#PageBegin 10 / AA -> the result is pages 10.AA, 10.AB, 10.AC..... You have to make sure that the number of pages does not exceed the number of digits given for the index. Example: The X1, X2…X50 terminal strips are used in a workspace. The index that is used is A, B, C… After the creation of Index Z, the next terminal strip is generated on a page with index AA, but, in the project tree, the AA page is sorted after page A and the sorting will be the following: A X1 AA X27 AB X28 B X2 etc.
#PageBreak <value>	A new page will begin, if the defined value changes. Example (List of Products): #PageBreak 140020, 140050 Each time the Function (140020) or Location (140050) changes, a page break is inserted and the list continues on a new page. The list can be sorted by the function in this way.
#LineBreak <value>	An empty line is inserted if the defined value changes.
#ColumnBreak <values>	The #ColumnBreak command allows you to switch to the next column if the given value changes (if a #PageBreak command exists and the value defined in there is the same defined in #Column Break, the page is changed). The same rules as for the #LineBreak command are valid.
#Pos. <value>	The placeholder #Pos. command enables you to get a line numbering (= consecutive numbers) in simple graphical lists like document list, product list, cable list etc. (line numbering is already possible in wiring list). The placeholder can be used together with a start value for the line number #Pos.[n], for example #Pos.[100] starts with value 100. If a start value for the line number is not found, the line number will start with value 1. The increment for the number always is 1. The line numbers cannot be used in terminal matrix, connector matrix, cable plan, cable terminal-row plan, products assembly, terminal plan, terminal row picture plan and connector plan lists.

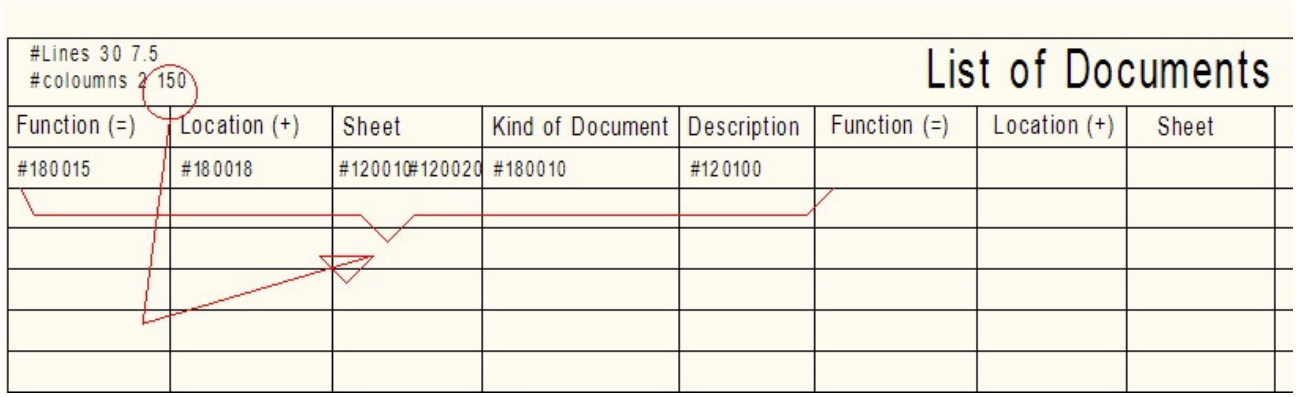
#OrderBy <values>	The graphical list is sorted according to the defined values whose codes (#number) are stated after the keyword #OrderBy. Example (List of Products): #OrderBy 140020, 140050, 160010 The List of Products is ordered, at first, by Function of the products in ascending order, then by Location of the products in ascending order, and then by component names in ascending order. You can sort the List of Products according to types #OrderBy 160040 or according to component names #OrderBy 160010.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

	Sorting in descending order is also possible: #OrderBy 140050 DESC, 160010 The List of Products is ordered at first by Function of the products in descending order, then by component names in ascending order. In Document List #OrderBy is not available
#Where [argument]	Records in the list are filtered according to the defined value. Example (List of Products): #Where [140050] = "+P1" AND [120010]>10 The List of Products will contain only components with Location "+P1" and placed on page number >10. Or #Where [160040] = "M10" OR [160040] = "M20" The List of Products will contain only components with types M10 and M20. Example (List of Documents): #Where [160040] <> " not used " The Product list contains all products without the ones with the type "not used". #Where NOT [160040] IS NULL or #Where [160040] IS NOT NULL or #Where [160040] <> " " The Product list contains all products without the ones with an empty type. Another example for use (Product list again): #Where Instr ([160010]), "M") >0 The component list contains only components with the code "M" in their name. Next example for use (Product list again): #Where NOT Instr ([160010]), "M") >0 The component list contains no components any more, that have the code "M" in their name. Example of use: Cable core list: The "Cable Type" is sometimes not entered (i. e. undefined, therefore you cannot filter on it, but this is necessary, since all entries are required, except for these with the value in the cable type: # WHERE (((IIf (IsNull ([160 200]), "", [160 200])) <> "Not Required")) Using the IIf function you can determine whether a term is true or false. If the term is true, IIf returns a value, if the term is false, IIf returns another value. You define the values, which are returned by Iif. The IIf function syntax uses the following arguments: DESCRIPTION OF ARGUMENTS expr required: a term that you want to evaluate. Truepart required. Value or term that is returned if expr is "true". falsepart required. Value or term that is returned if expr is "wrong". Thus, IIF returns the value " ", if 160200 has no entry (and thus

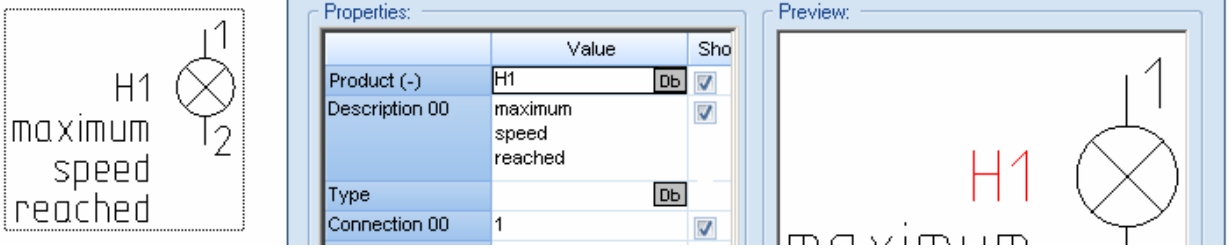
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

	we get an entry) and the value for the cable type, when 160 200 is an entry. Using Where you can give out only that, what is not equal to the term "Not Required". In Document list there are specific rules, refer to the specific list to get to know about them. In Terminal Matrix and Terminal Plan, Connector Matrix and Plan, Terminal Row Picture Plan, and Cable Plan, this attribute is not available. In other lists might exist several restrictions.
#Columns <number of columns> <offset for next column>	It is possible to output more than one column in multiple graphical lists that are filled line by line: Documents list, Product list, Part list, Cable list, Cable wire list, Terminal list, … In Terminal Matrix and Terminal Plan, as well as in Terminal Row Picture, Cable Plan and Products Assembly this option is not available.



	If two or more columns have to be generated, the page template must contain a text with "normal" attribute and the following content: #Columns <number of columns> <offset for next column> For example, #columns 2 150 like in the example shown above. The second column needs to contain the geometry and the texts for the headline but no placeholder IDs such as #180015. The #ColumnBreak command allows you to switch to a new column if the defined criteria is executed (if #PageBreak exist, it has a higher priority). Syntax: #ColumnBreak <Value> The rules are similar to the ones valid for the #PageBreak. Additionally, there are commands that allow you to define in which direction will be filled the columns. #RightThenDown: place the first entry in the first line, place the next entry in the first line of the next column #DownThenLeft (=default if nothing is defined): fill all lines in the first column, then start with line 1 in the second column
Output multiline text in a single line in lists	If, for example, the text "description 00" is to be split in three lines in a component to fit the space available in the circuit diagram, it is possible to display this text in one line in the product list.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



	If, for example, the text "description 00" is to be split in three lines in a component to fit the space available in the circuit diagram, it is possible to display this text in one line In the product list. In the Terminal Matrix and Terminal Plan, as well as in the Terminal Row Picture, Cable Plan, Connector Matrix, Connector Plan and Products Assembly this option is not available. To output multiple lines in one single line, the page templates must be enhanced in the following way: The ID for the "Description 00" text is 160030. If in the template for the product list is present the #160030 $SingleLine Placeholder, the text is displayed in one line maximum speed reached If no separator is defined, a blank one is used after each linebreak. If a separator is defined, it is used. If in the temp0late for the product list is present the #160030 $SingleLine [-] placeholder, the text is displayed with a – as a separator. Maximum-speed-reached.
Use word wrap in an expression	It is possible to use a line break inside a text added to a simple graphical list, to make the text fit a column. To use the wrapping the page templates of the simple graphical list be defined in the text that contains the ID. Syntax for wrapping is: %wt(<number of characters>). If no maximum number of characters is defined, then wrap is ignored. All other column format definitions are valid. The wrap text algorithm is applied to the text after all other formatting is made. Examples: 1) if there is a column with ID #120100 (Page Description 1), the wrapping is defined like this: #120100 %wt(10) -> the wrap is made after 10 characters. 2) if content of a column is made from several Ids (=a formula is used) like #fu[#120100][,][#120110], the wrap definition must be placed inside the information about the first ID to use: like this #fu[#120100 %wt(7)][,][#120110] The wrapping is applied on the entire formula text.
Output multiple values in one field in lists	In the simple graphical lists like Document List, Product List etc. it is possible to combine several values in one entry. For example for components with more than one type the product list can now be

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

	generated in a much nicer way. The combination of several values is done using the #FU command. This command uses the following syntax: #FU[#<1st ID>][Separator][#2nd ID][;][#3rd ID] Examples: - Combine 2 types in the Product List: (Separator ;): #FU[#160040][;] [#160041] - Combine Page description 1 and 2 in the Document List (Separator space): #FU[#120100][ ] [#120110]
The number of digits and the length of the text can be controlled.	Some formatting attributes have been introduced to allow the formatting of information displayed in the graphical lists (none of these formatting attributes can be combined with another one of these attributes). In the Terminal Matrix and Terminal Plan, as well as in the Terminal Row Picture, Cable Plan and Products Assembly this option is not available. Attention: only one of this formatting attributes can be used. Numerical values: Control the number of decimal places Add to your template a text placeholder with "Normal" text attribute that contains the command #<ID of text attribute> %.<n>f or #<ID of text attribute> %d Add this if three decimal places are necessary for the amount (ID 180040) in the part list. The result for the amount in the part list is 5.004. #180040 %d Add this no decimal places are necessary for the amount (ID 180040) in the part list. The result for the amount in the part list is 5 For example: #180040 %.3f Add this if three decimal places are necessary for the amount (ID 180040) in the part list. The result for the amount in the part list is 5. Control the decimal places and add static text Add a text placeholder with "Normal" text attribute to your template that contains the command #<ID of text attribute> %<n>.f<text> For example: #160101 %.2fmeter Add this if two decimal places are necessary for the length of the cable (ID 160101) in the cable list and the text "meter" must be inserted after the value. The result is 5.45 meter. Use full integer formatting Add to your template a text placeholder with "Normal" text attribute that contains the command #<ID of text attribute> %0<n>d For example: #120010 %03d Add this if three integer values are necessary for the length of

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

	the page number (ID 120010) in the Document list. If are present the pages 5, 11, 123 and 444, the result is the following: 005 or 011 or 123 OR 4444 The #<ID of text attribute> %0<n> d command states that the 0 is added to the value as often as necessary to reach the minimum length of the string defined by <n>. Is a value exceeds the minimum length it does not change. Text strings Control the length Add to your template a text placeholder with the "Normal" text attribute that contains the command #<ID of text attribute> %<n>s For example #160030 %30s Add this if only 30 characters fit into the field available for the "Description 00" text (ID 160030) in the Product list. The text is truncated after 30 characters. If, for example, are used 50 characters, only 30 are displayed.

Generate all desired graphical lists in one step

Within *SEE Electrical*, you are able to generate all desired graphical lists in just one click.
Using internal codes within the graphical lists, you can define a "chain reaction" at graphical lists generation. The internal codes also allow you to specify consecutive page numbering valid for *all* lists, e.g. the first document obtains page number 1, the next - 2 and so on, until completed. This enables more flexible handling of graphical lists.
The following attributes must be specified within the list:
#NextList List ID

Example :
#NextList 3100
There are no brackets [ ] allowed.
For this example, the next list generated will be 3100. This code must be inserted in the first list that you generate. The first list starts a "chain reaction", i.e. this code will automatically call and generate the next kind of graphical list (in this example, list type 3100).

The IDs for the types of graphical lists are as follows:

2000 - Other documents
3001 - Documents list
3011 - Products list
3020 - Terminals list
3025 – Connector list (only *Advanced*)
3026 Connector Pin list (only *Advanced*)
3030 - Cable list
3031 - Cable-core list
3050 – PLC I/O list
3060 - Wires list
3100 - Parts simple list
3101 – Parts list, Simple
3102 - Spareparts list (from *Standard*)
3103 - Spareparts list, Simple (from *Standard*)
3104 - Terminal matrix (from *Standard*)
3105 - Cable plan (from *Standard*)

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

3108 - Connector matrix
3106 - Cable terminal row plan (only *Advanced*)
3110 - Parts simple F&L sorted
3112 - Parts F&L sorted (only *Advanced*)
3220 - Products assembly (only *Advanced*)
3225 – Aspect Functions (only *Advanced*)
3226 – Aspect Locations (only *Advanced*)
3235 - Cables list F&L Sorted (only *Advanced*)
3236 - Cable core list F&L Sorted (only *Advanced*)
3270 - Terminal plan (only *Advanced*)
3271 - Terminal row picture (only *Advanced*)
3273 - Connector plan
3280 - Multicores list (only *Advanced*)
3285 - Multicores wires list (only *Advanced*)

PageBegin [List ID]
Example:
PageBegin *[3000]* –
The brackets [ ] must be used!
Page numbering of the current list continues the page numbering from the last page number generated in the list with ID 3000.

PageBegin [List ID] $
The brackets [ ] must be used!
Example: [3000] $ - Pages have the numbers in the function groups in list 3000: consecutive page numbering independent from the type of the list in which the data is inserted.
PageBegin <page number> (for ex. 20)

Example: 20 - The page numbering starts at page 20.
PageBegin <page number> % (for ex. 20%)
Example: 20% - Page numbering starts at page 20 for *each* function group.

P.3. GENERATE COVER SHEETS FOR GRAPHICAL LISTS

All graphical lists can generate a cover sheet if necessary.
This is done with the text #CoverSheet <"template name for cover sheet"> (text attribute "normal" text) inside the page template of the graphical list.

Example for document list:
#CoverSheet "Cover Document List".
The page template for the cover sheet *CoverDocumentlist.TDW* must be located in *…\templates* folder.
In the properties for the graphical list is defined the template to use for list generation and not the one to use for the cover sheet.

The cover sheet for the list is always loaded. Inside a coversheet no data source will be used. So the coversheet is a static drawing where only the default objects are shown and project attributes are read.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Q EXTERNAL DATA TRANSFER

Q.1. DATA TRANSFER THROUGH DWG/DXF/DXB FORMAT

Data transfer between various CAD-systems is possible through the *DWG/DXF/DXB* format. However, it is not possible to transfer data about electro-technical logic while transferring data through these formats.

Exercise 16-1: Import data into *SEE Electrical*.
Create a new empty page first.

1.CA **File**
2.CO **Open**
3.CO **AutoCAD DWG/DXF/DXB Drawing**…
5.> Files of Type
6.> AutoCAD DXF files (*.dxf)
Select the desired type of file.
7.> Look in
Select the folder where the file is located.
8.> Select the file.
9.> "Fit contents to Page"
The properties of the imported drawing have to be adjusted. The option must be ticked.
10.> Delete Page before import
The available drawing (i.e. the empty standard sheet) must be deleted. The option must be ticked.
11.> Open
The file has been imported and is displayed as a drawing.
You can edit it now as desired.

Exercise 16-2: Save data within *DXF* format.

1. Switch to any page in the workspace that you want to save within a *DXF*, *DWG* or *DXB* format.
2.CA **File**
3.CO **Save as**
4.CO **AutoCAD DWG/DXF/DXB Drawing**…
5.> Save as type
6.> DXF v2004 File format (*.dxf)
Select the desired type of file.
7.> File name
8.> <name>
Type the name of the file.
9.> Save in
10.> Select the folder where the file must be saved.
11.> Save
The file has been saved.
Using the same approach, you can save DWG/DXF/DXB files in all pages of your project by activating the **File** □ **Save as** □ **AutoCAD DWG/DXF/DXB Workspace…** command.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Q.2. PIXEL IMAGES TRANSFER

You can insert pixel images (JPG, BMP, etc.) in *SEE Electrical* drawings.

Exercise 16-3: You can import a logo as a Bitmap object.

1.CA **General**
2.CO **Picture** (**Insert** panel)
The following formats are supported: *PCX*, *JPG* and *BMP*.
3.+ Select the first point of the frame, where the Picture is to be inserted. The frame is defined as a rectangle by two opposite corner points.
4.+ Select the second point of the frame for the Picture.
5. Select the desired file.
6.> Open
You have just inserted the Bitmap object into the drawing.
You can modify the Bitmap object.
7. Click the icon to enable trackers on selected elements.
8.+ Left-click the bitmap object:
You can see the trackers on each corner point and between them.
9.+ Use the trackers to change the size of the bitmap object.
10.+ Click within the bitmap object, hold the left mouse button and move the object to the desired position.
11. Click the icon again to switch the trackers off.
12. To rotate the bitmap select it and use the **Rotate** pop-up command to rotate it at 90 degrees.

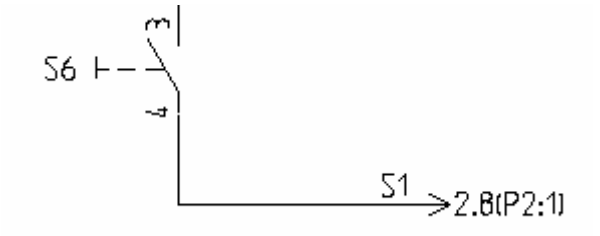
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

R PROCESSING A WORKSPACE

(Standard)

R.1. ADDITIONAL OPTIONS FOR CROSS-REFERENCES

Exercise: Show targets for cross-references.



1. Select the cross-reference, which you want to show the target for.
2. Right-click to open the pop-up menu..
3. M. **Properties**
4. # Change the default setting for the **Show target** value from **Off** to **On.**
   Type in the desired text.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

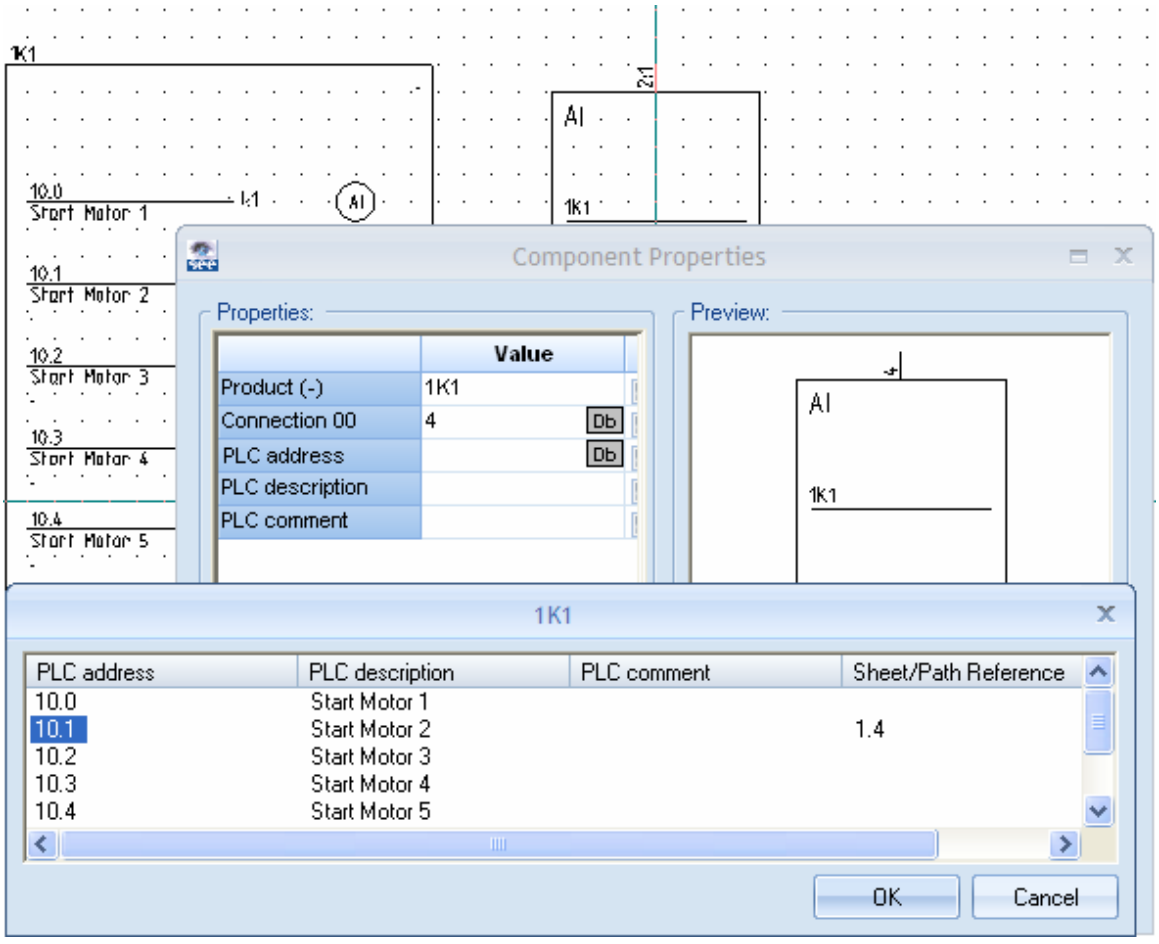
R.2. PLC FUNCTIONALITIES IN THE STANDARD LEVEL

R.2.1. USING THE PLC DATABASE

In *SEE Electrical Standard* level, there is a possibility to make sure you assign only those PLC addresses used in the rack, if there is one.
To do so proceed as follows: when giving the name and the PLC address to a PLC I/O, give the name first. After that you can click the Db button in the "Connection" line or in the "PLC address" line.

Connection 00	4	Db
PLC address	?	Db

After that a window appears, that shows all the PLC addresses defined in the rack. If PLC description or PLC comment is defined in the rack, you see the appropriate information.
An entry in the column "***Code/Cell reference***" shows, this I/O is already used.

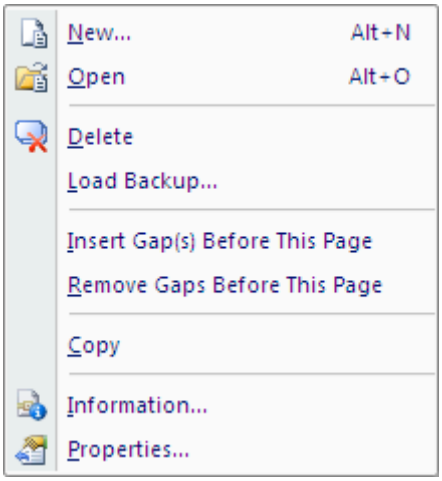


Review
If a Rack is already placed, it will be checked while inserting the inputs/outputs if the address is already available, and if the number of connections at the input/output complies with the number of the connections at the Rack input/output. An error message can appear.
In the Check PLC Connections database list, such errors are documented if you do not correct them.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

R.3. ADDING AND DELETING PAGES

If you select a page in the ***Workspace Explorer*** and you right-click with the mouse, a pop-up menu appears, allowing you to add pages before the current page (**Insert Gap(s) Before This Page**) or remove empty pages before the current page (**Remove Gaps Before This Page**).



If you use a page based numbering, you can choose whether to change the component names or not.
The functions are available for each kind of plans, as well as for graphical lists. After generating terminal matrices, it is possible, for example, to start on page number 300.

R.4. COPY SINGLE PAGE IN THE SAME WORKSPACE

If you select a page in the ***Workspace Explorer*** and you right-click with the mouse, a pop-up menu appears, allowing you to copy the current page. You can later paste it in the active workspace. When the command is activated, the ***Page information*** dialogue appears. It contains all page information texts existing in the copied page. Assign the new page number, modify any page information and click **OK**.

Component names are either automatically adapted, or you are asked to confirm their names if the "*Component Numbering*" is set to "*Free*". References are automatically updated.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

S EASY EDITING IN DATABASE LISTS

(standard)

In *SEE Electrical standard*, there are editors available in the *Database* lists.
You can change data via these Editors. The changes are saved in the circuit diagrams.

Exercise 17-1: Make changes in the Product Editor.

1. Select the row of the record you want to change, for example the component Q1.
   The component information is displayed in the pane on the right.
2. In the right pane, select the row you want to change, for example Circuit breaker.

	Text Value
Product (-)	Q2
Description 00	Circuit breaker
Type	3VE1010 Db
Connection	1
Connection	2
Connection	3
Connection	4
Connection	5
Connection	6

3.# Circuit breaker
Type in the desired text.
Click the next component you want to change.
Look at the page of the circuit diagram, where the changed components are located. The modifications are visible.
If you have changed the way of automatic numbering of the components or you want to rename all the components and the Automatic numbering has been selected (=> ***Circuit diagram properties***, ***General*** tab, ***Component numbering***), you can rename them by executing the **Renumber all components on all pages** command in the Product editor.

If you inserted or removed terminals, you can change the terminal number and sorting via the terminal editor executing the Renumber all shown terminals command found in the pop-up menu.

Through the pop-up menu, in the Contact list is available the **Renumber all contacts on all shown components** command, which allows you to renumber all the new contacts.
The renumbering of the contacts can be made only if the contactors / relays / components with auxiliary contacts have a valid channel definition. This is necessary because the connection information is extracted from the channel definition even by renumbering.
The renumbering starts on the first page, then continues page after page and on each page it begins from the upper left to the lower right corner.
If you use a filter for example for one page, the contacts of all contactors / relays / components with auxiliary contacts, located on the page, will be renumbered, i. e the master symbols are always decisive. If you filter on a single contactor / relay / component with auxiliary contacts, only the contacts on this will be renumbered.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

T CONTACT MIRROR AND TYPE DATABASE

(Standard)

In *SEE Electrical standard,* you can choose to add cross references between a relay coil and its related contacts: using either a contact mirror or a contact cross.

Exercise 18-1: Check the circuit diagram properties for the project. A contact mirror instead of a contact cross must be used in the future. (You can save this setting in your workspace template.)

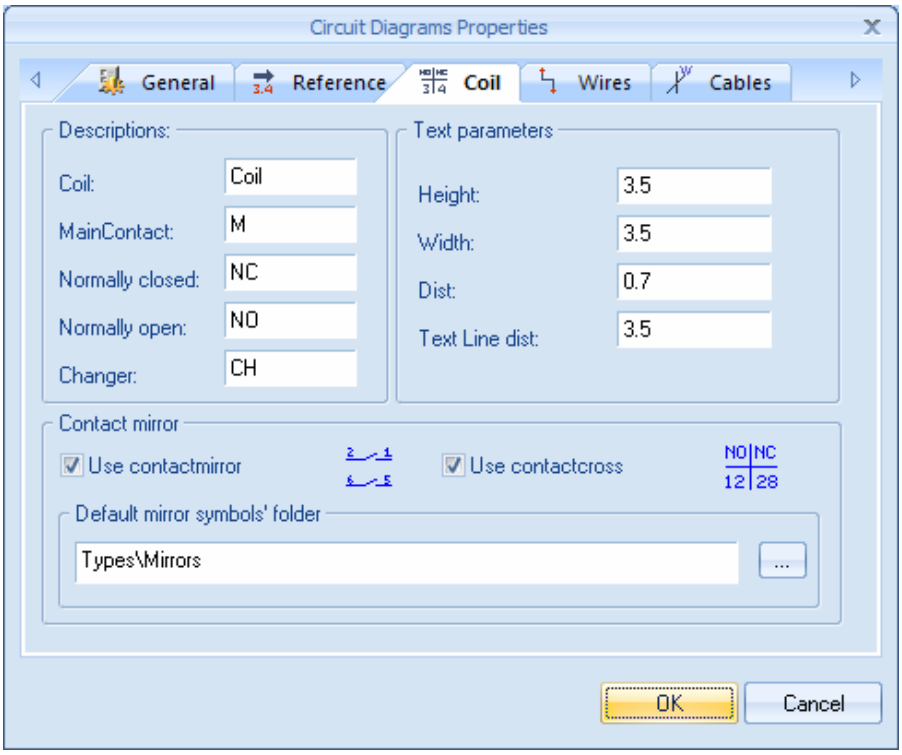
1. Click *Circuit diagrams* in the ***Workspace Explorer***.
2. Right-click with the mouse
3CO **Properties**…
   The ***Circuit Diagrams Properties*** dialogue box appears.
4.T Coil
   Select the ***Coil*** tab.
5.> Use contact mirror
   Tick the option.
6.> **OK**

Exercise 18-2: Use contact mirror if the article information is not defined.
A symbol database with the symbol for the contact mirrors has to be defined in the ***Coil*** tab of the ***Circuit Diagrams Properties*** window. The first symbol in the symbols folder that has the same symbol ID as the already inserted contact is used for the building of the contact mirror. For example, if, in the symbols folder you have two symbols with ID for the main contact – one single pole and one three pole, the first one is used. Additionally you have to activate the "***Use contact mirror***" option.

1. Click *Circuit diagrams* in the ***Workspace Explorer***.
2. Right-click
3.CO **Properties**…
   The ***Circuit Diagrams Properties*** window appears.
4.T ***Coil***
   Select the ***Coil*** tab.
5.> Choose the *Default mirror* symbol folder to use.
6.> Choose the **Types** library and in there the **Mirrors** folder.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

6.> **OK**



If you want to use article data with your coils, follow the rules defined in the next chapter.

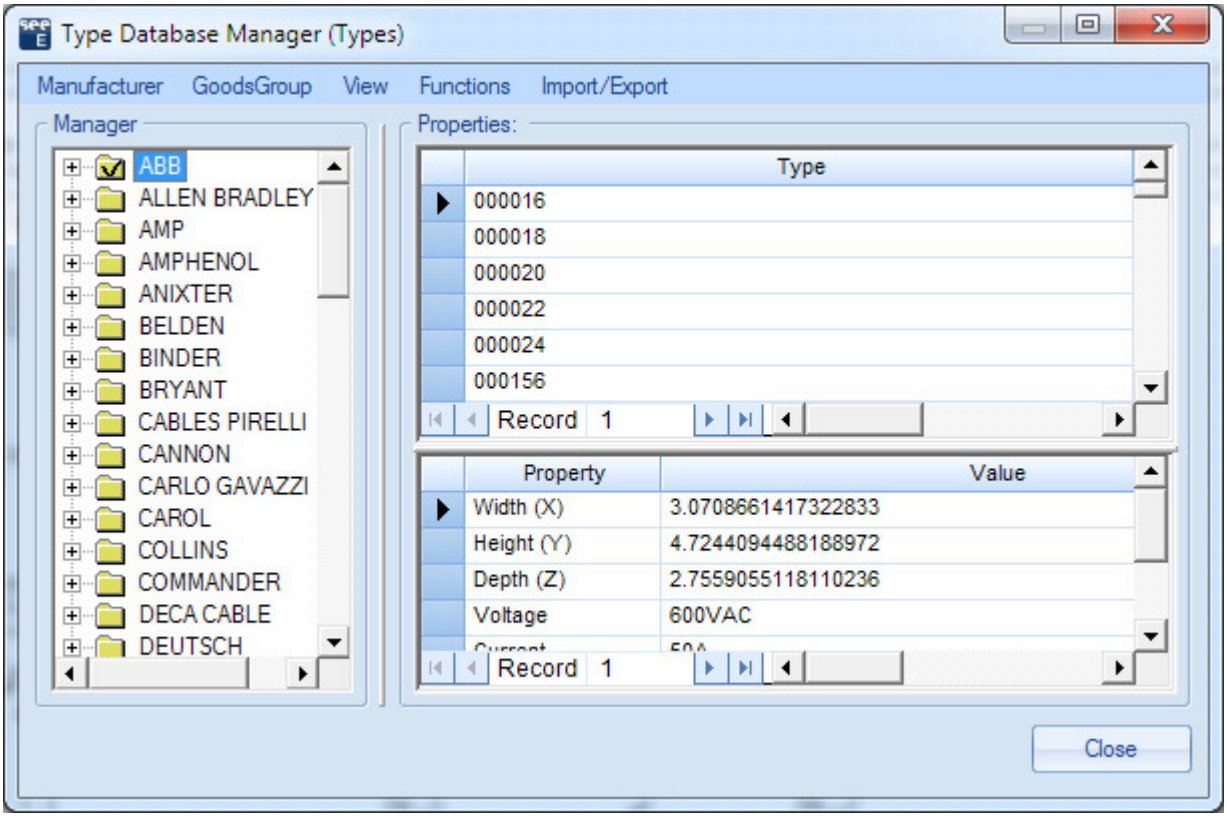
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

T.1. MANIPULATING THE TYPE DATABASE

If you decide to use the article data on a coil, the contact mirror provides information about contacts available in a relay coil, and about contacts already used. The contact mirror can be created correctly only if it is known which contacts are available in the coil, i.e. what type the coil possesses.
The way of constructing the contact mirror is defined in the so-called *Channel definition*. This can be performed in the Type database.

Exercise 18-3: Create a new Manufacturer.

1.CA **Functions**
2.CO **Database** (**Types** panel)



The ***Type Database Manager*** window appears.
This window is composed of three areas. You can move the borders of the areas and the borders of the columns in the right panes.
Adjust the ***Type Database Manager*** window as desired. Do the next steps as described.
Create the new manufacturer "*Training*".

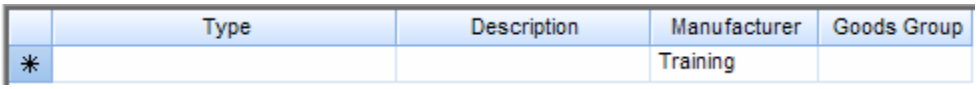
3.M Select the **Manufacturer** menu.
4.M **Add**
5.# Training
Type in the name of the new manufacturer, i.e. **Training**.
6.> **OK**
The manufacturer has been created.
Select the newly created "*Training*" manufacturer in the left pane of the ***Type Database Manager*** window.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

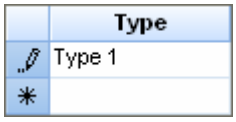
Exercise 18-4: Define a new type of a relay-coil.

Types must be unique, i.e. one type designation must appear only once.

1.M Click in the field under the ***Type*** column in the top right pane.



2.# Type 1
Enter the new type designation.



3.# Relay coil AC
In the "***Description***" field, you can write the description of the new type, for example Relay coil AC.

4.> You can choose the desired manufacturer from the list by clicking within the *Manufacturer* column. The "*Training*" manufacturer is already available.

5.> Click within the *Goods Group* column. The icon appears. If you click this icon, a list with the available goods groups opens. Select the desired goods group, in our case - "Auxiliary Contactors".
The goods groups are used for structuring the goods information. For example, in *SEE Electrical Standard* and *Advanced* levels***,*** you can enter data about types for all kinds of components, i.e. switch gears, terminals, relay-coils etc. But, when you assign a type to a component, only the relevant type is to be shown. The goods groups enable this pre-selection.

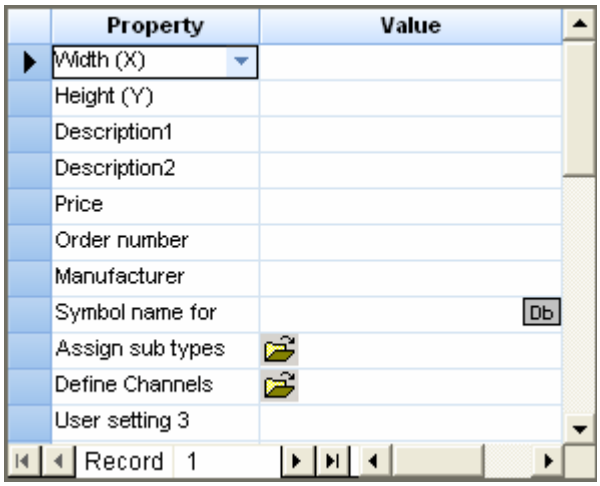
6.> Expand the "*Training*" manufacturer.
The goods groups assigned to this manufacturer are shown, i.e. "Auxiliary Contactors".

7.> Expand the "Auxiliary Contactors" goods group.
The new type appears now in the Type Database Manager, too.

8.> Select the type you wish to assign information to, by clicking it in the Manager pane in the left part of the window.

In the bottom right pane of the ***Type Database Manager*** window, all the default properties of a type are displayed. (The default properties can be changed via the **Functions** □ **Settings** command, which opens the ***Settings*** window.)

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

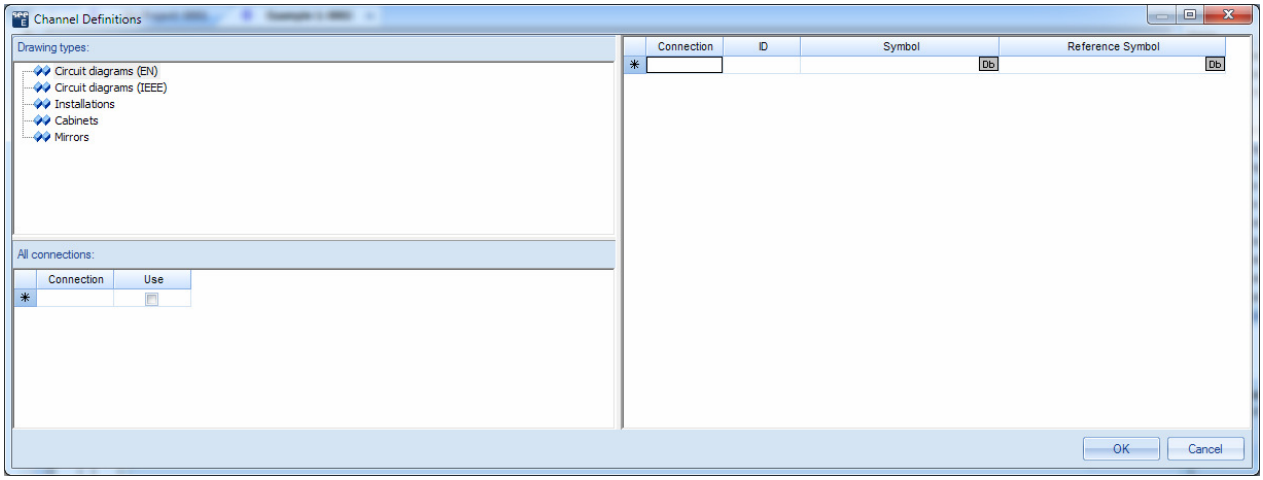


9.> Click the *Value* column for the respective properties.
Type in the following values:
Width: 100 (used in module Cabinets)
Height: 100 (used in module Cabinets)
Description 1: AC relay-coil
Order number: 123A

Exercise 18-5: Define channels for your relay coil type.

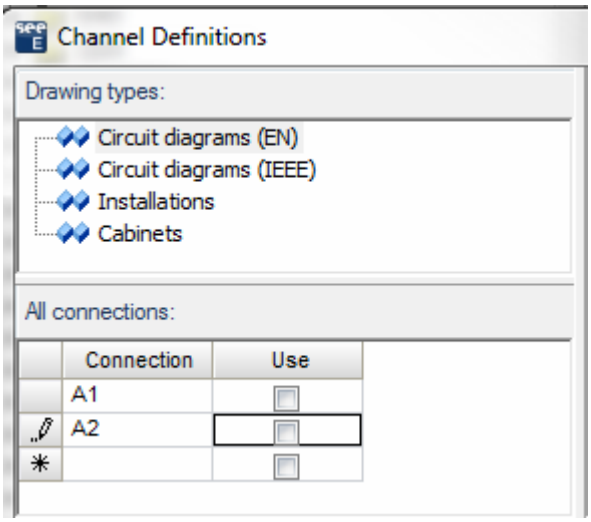
A relay-coil or a component with auxiliary contacts needs the information about the contact mirror that must be used. When a contact mirror must be shown under the relay-coil symbol, this property is essential. It is also necessary for a component with auxiliary contacts if an automatic contact numbering is to be performed.

1.> Click the icon in the ***Value*** column for the *Define channels* property.
The ***Channel Definitions*** dialogue appears. Perform the channel definition.

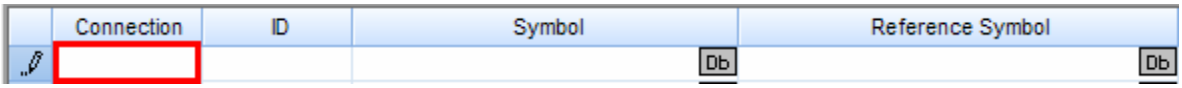


2.> Select the *Circuit Diagrams (EN)* drawign type.
3.> In the ***All connections*** area, insert the connection texts A1 and A2

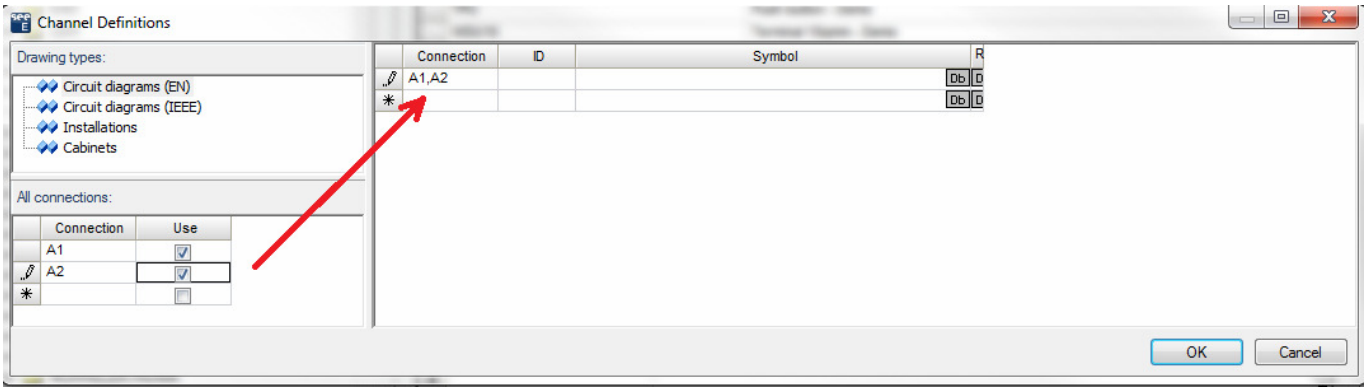
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



4.> Click in the "***Connection***" field:



5.> Check the "***Use***" check-boxes in the ***All connections*** area.



6.> ID
7.# Relay Coil
Select the *Relay Coil* ID from the pull-down list.
8.> Symbol

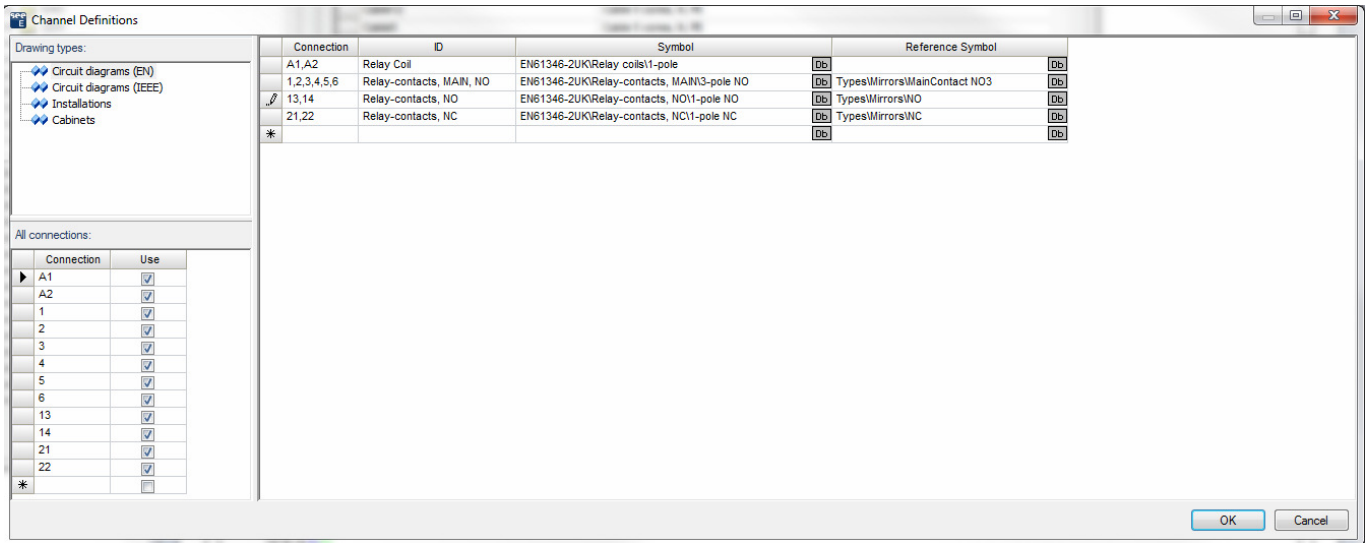
Click on the Db button within this field. Select the desired symbol from the *Relay Coils* folder of the *EN61346-2UK* symbol library. The path to the selected symbol appears in the *Symbol* field.
9.> In the same way, in the ***All connections*** area, insert the connections: 1, 2, 3, 4, 5, and 6. Repeat steps 3 and 4.
10.> Click the "***Connection***" field of the second line.
11.> Check the "***Use***" check-boxes in the ***All connections*** area for the connection texts 1, 2, 3, 4, 5 and 6
12.> ID
13.# Relay-contacts, MAIN, NO
14.> Symbol

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

15. Click on the Db button within this field. Select the **3-pole NO** symbol from the Relay-contacts folder, MAIN folder of the EN61346-2UK symbol library
16.> Reference Symbol
Click on the Db button within this field.
17.> You can select the contact mirror symbol from the symbol databases again.
Open the *TYPES* database, and then open the *Mirrors* folder by double clicking on it and select the **Main Contact NO3** symbol.
18. Repeat steps 4 to 17 for the NO auxiliary contact.
19.# 13, 14
Type the contact numbers of the first NO contact.
20.# Relay-contacts, NO
Select the *Relay-contacts, NO* ID from the pull-down list.
21.> Symbol
Click on the Db button within this field. Select the desired symbol from the Relay-contacts, NO folder of the EN61346-2UK symbol library.
Click on the Db button within this field.
22.> Reference Symbol
Click on the Db button within this field
You can select the contact mirror symbol from the symbol databases again.
Open the *TYPES* database, and then open the *Mirrors* folder by double clicking on it and select the *Contact NO* symbol.
23.> Connection
24. Repeat steps 4 to 17 for the NC auxiliary contact
14.# 21, 22
Type the contact numbers of the first NC contact.
15.# Relay-contacts, NC
Select the *Relay-contacts, NC* ID from the pull-down list.
16.> Symbol
Click on the Db button within this field. Select the desired symbol from the Relay-contacts, NC folder of the EN61346-2UK symbol library.
17.> Reference Symbol
Click on the Db button within this field once again.
18.> You can select the contact mirror symbol from the symbol databases again.
Open the *TYPES* database, and then open the *Mirrors* folder by double clicking on it and select the *Contact NC* symbol.
19.> **OK**

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Finish the channel definition.



Exercise 18-6: Copy the type you have just created and copy a different channel information..

1.> Click the type in the type database

2.>

	Type
▶	Type 1
*	

2.M **Functions**

3.M **Copy Selected Type**

A copy with the type name "*Copy of <type>*" appears, now it is "*Copy of Type 1*".

Now copy a new channel definition for this type.

4. Make sure the type to add the channel definition to is selected.

5. In the **Functions** menu of the ***Type database manager*** window choose the **Copy channel from other type** command.

6. Select the type to copy the channel information from like you select a type normally. (Only one type can be selected here.)

7. Terminate your selection. Doing this the channel information is copied .

4.> You can close the ***Type Database Manager*** window by clicking the **Close** button.

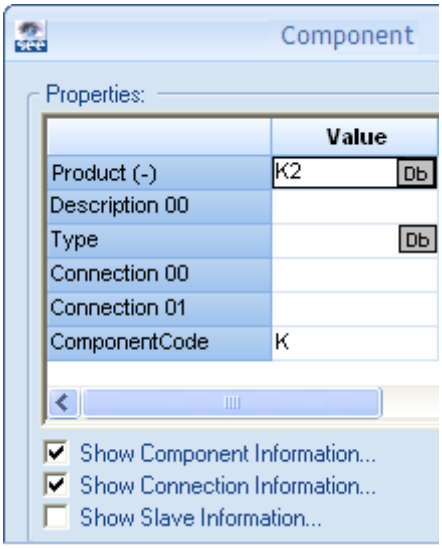
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

T.2. USING TYPES IN THE CIRCUIT DIAGRAMS

Use the newly defined type in the current circuit diagram.

Exercise 18-7: Switch to page 2 of your training workspace, and define a type for the relay coil K2.

1.+ Double-click the relay-coil.

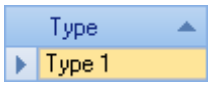


You can fill in the type in the Type field using the keyboard or you can select the type from the type database.

2.> Click Db.
The ***Type Database Browser*** window opens.

3.> Open the Training manufacturer.

4.> Select the Type 1 by clicking the field on the left of it.



5.> Click the **Select** button.
The type is selected and appears in the right pane of the window.

6.> Close the ***Type Database Browser*** window by clicking on the ✕.

7.> **OK**
Close the ***Component Properties*** dialogue box.
The contact mirror appears.

Exercise 18-8: Assign a type to coil K3.

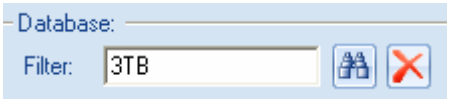
You can filter in the type database.

1.+ Double-click the K3 coil.

2.> Click Db in the "***Type***" field.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

3.> Filter



4.# 3TB
Click the icon to confirm.
All the types containing "3TB" in their name are displayed.
Please use types that provide a suitable number of contacts.

5.> Select a type by double-clicking in the field left to the Type, in this example 3TB46170BB4.
The type is selected.

6.> Close the ***Type Database Browser*** dialogue by clicking **OK**.

7.> **OK**
Close the ***Component Properties*** dialogue box.

Exercise 18-9: Filter the coil types in the Type Database according to the number of used contacts

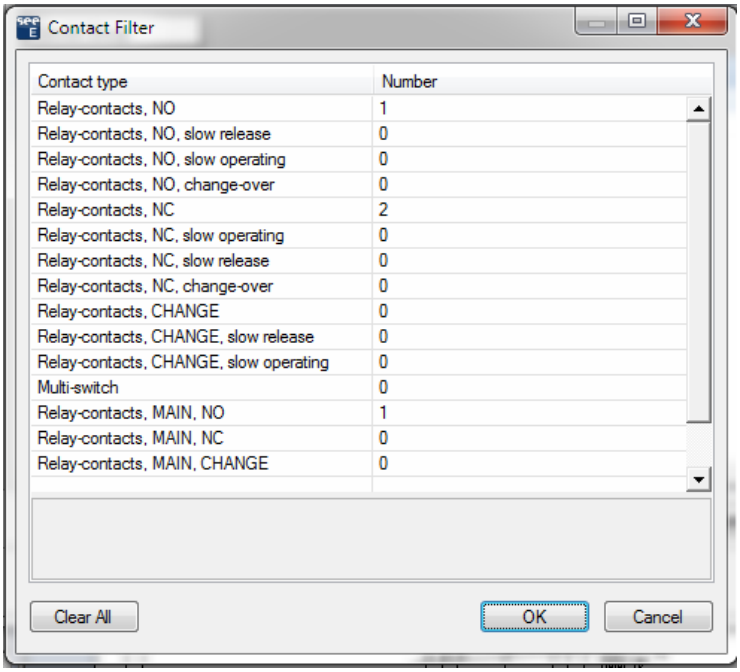
1.+ Double-click the K4 coil.

2.> Click Db in the "***Type***" field

3. + Tick the "***Activate***" option in the ***Contact Filter*** area of the ***Type Database Browser***.
After the filter is activated, the list will contain only the necessary number of contacts.
You can additonaly add more contacts to the filter manually.

4.> Define the desired filter – the base of the filter is the number of contacts already present in the circuit diagram.

5.> To define more contacts, use the **Define** button..



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercise 18-10: Assign types to the two motor protective switches on page 1.

1.+ Double click one of the motor circuit breakers.
The ***Type Database Browser*** window opens.

2.+ Click in the **Type** field.

3.> Manufacturer..

4.> <Manufacturer> - Select as manufacturer **IGE+XAO.**

5.> Click in the field, where you want to group, e. g. **Goods group** and drag it with the mouse button pressed in the "**Drag a column header here to group by that column**" position. If necessary, drag another field in this position.

6.> Through the "**Collapse**" button you can close the folder of all grouping at once
Close the ***Component Properties*** dialogue box.
The contact mirror appears.

7. If you want to cancel a grouping, click on the appropriate field and drag it with the left mouse button pressed in the area of the column heading. Release the mouse button, when you can see red arrows.

8. <Type>
Select the **3VE1011** Type for the motor circuit breaker. You can find the type in the **Q** goods group.
Select the type through double click on the area in front of the type.

9. Close the ***Type Database*** window with the button.

10. OK.
Leave the ***Component properties*** window.

Exercise 18-11: If contactors or components with auxiliary contacts have a type with a channel definition, the contact numbering is performed automatically. If no free contact of the relevant type is available in the component, an error message appears.

1.+ Place a new contact, and assign it to an available contactor or a motor protective switch.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

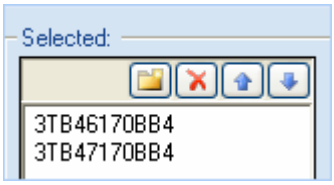
T.3. AUXILIARY CONTACTS INSIDE ADD-ONS FOR COILS AND SUBTYPES

Components do not always consist of only one element, they are often assembled from several elements or elements can be added to an article.

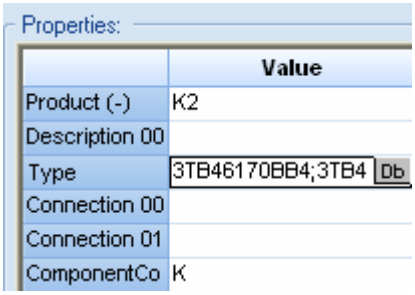
Fuses are typical examples of components assembled from several articles. They always consist of a socket, a fuse link, a screw cap and a fitting screw. In *SEE Electrical* these parts are called subtypes.
For a relay coil, an add-on is needed sometimes, but sometimes not.
Therefore, *SEE Electrical* offers two different approaches.
All of the articles must be stored in the type database.

Exercise 18-12: Assign an add-on for a relay coil.

1.+ Double-click the relay coil.
2. Go into the type database and select a second type.



3. Close the ***Type Database Browser*** dialogue by clicking **OK**.

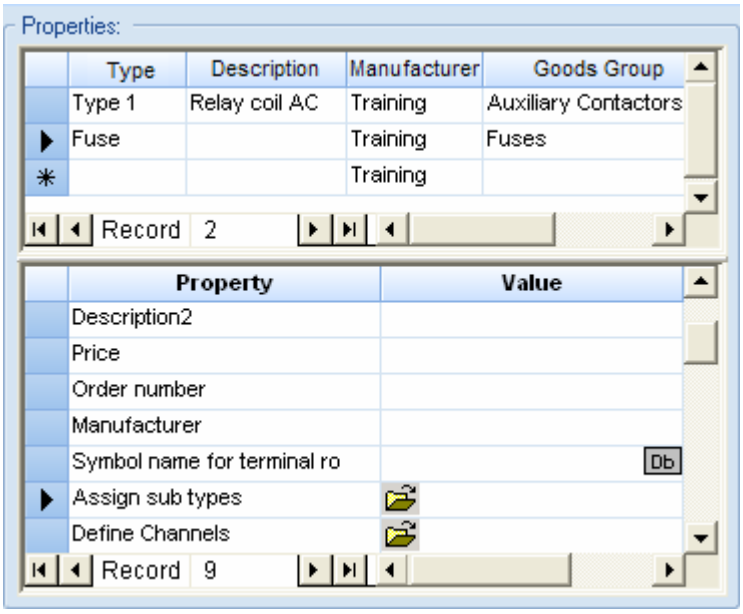


3. Both types are separated by a semicolon in the "*Type*" field of the ***Component Properties*** dialogue box.
You can assign up to 10 types to a component.
Look at the Products list. A column Type 02 with the second type is created. Look at the Parts list afterwards. The types remain on single lines there.
Add-ons must be added to relay coils always using this approach.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercise 18-13: Define and use a type for a fuse.

1.+ Insert a fuse into your drawing. Double-click the fuse.
2. Go into the type database and create a type "*Fuse*".
The needed elements for the fuse are available. Otherwise, define them before you create the fuse type.
3. In the "***Property***" column, you can find the Assign sub types property.



4. Select Assign sub types.
5. You can choose all the types needed as elements for your fuse.
If several sockets or melt inserts are needed, choose the appropriate type several times.
6. Close the ***Type Database Browser*** dialogue by clicking **OK**.
Define then the next fuse type in the same way.
7. Assign the type to the fuse.
Look at the *Products* list. For the component "*Fuse*", only one type is recorded in the list of products, as well as in the Spare parts list.
In the Spareparts, simple list, each element of the fuse appears on a separate line.
Look in the *Parts* list. You only find the main type, too.
In the Parts, simple list the parts are on separate lines.

Once determined, the types for components with subtypes must not be changed again.
If a fuse sometimes has a splash guard and sometimes not, the type for the splash guard can be defined as a second type for the fuse.
If you need a 16A fuse now, but you need a 4A fuse later, define two types with different subtypes, etc.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

U ADDITIONAL HINTS ABOUT COMPONENT MANAGEMENT IN THE TYPE DATABASE

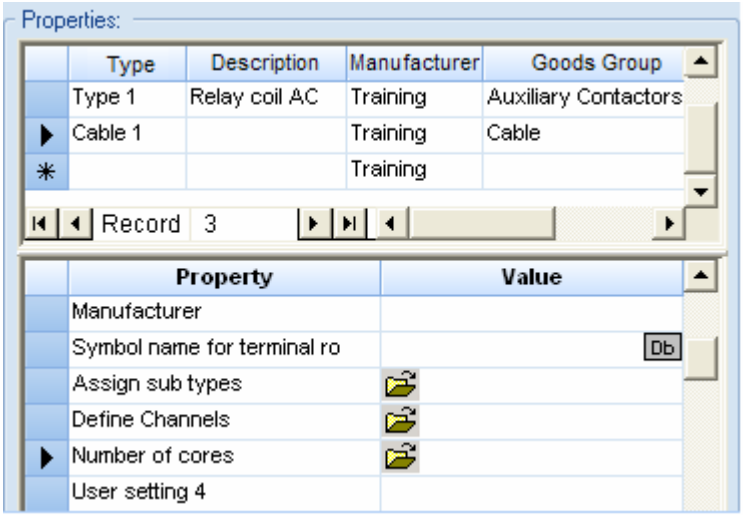
(Standard)

U.1. CABLE MANAGEMENT

SEE Electrical can manage information about Cable-cores in the type database and Cable-cores used in a cable.

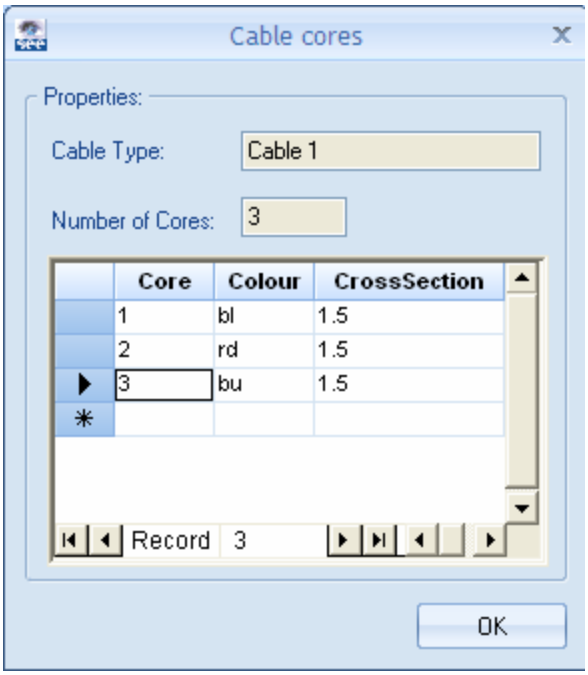
Exercise 19-1: Define a cable type.

1. Open the type database and create a cable type as usual.
2. In the ***Property*** pane, find the *Number of cores* attribute.



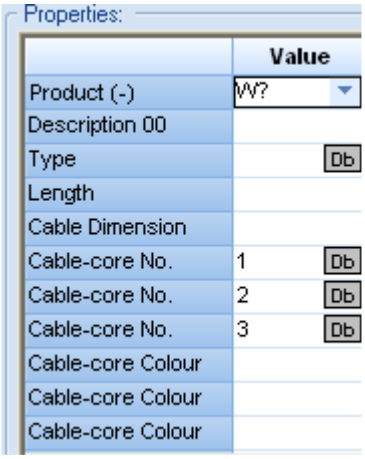
3. Click the icon and define a colour and a cross-section for the cable-cores.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



4. Click **OK** to finish the definition.
5. Close the type database.

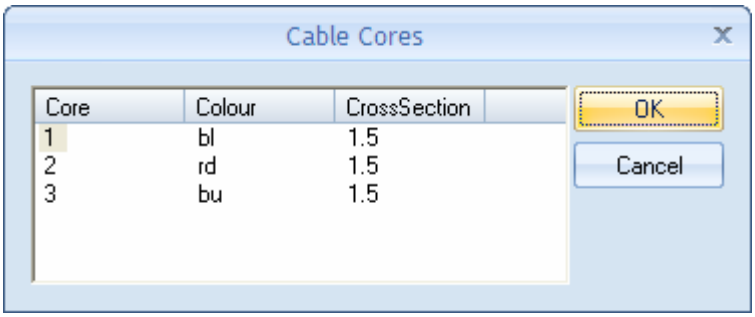
Exercise 19-2: Draw a new cable. Use the defined type.



Assign the newly defined type to the cable. The number and cross-section for the cable-cores are filled in automatically as defined in the type database.
If you wish to change the order of the cable-cores, delete all the core numbers, then click the Db button in the first "*Cable-core No.*" line.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

A database opens, where all the free cores of the current cable are displayed. You can choose a core here.



Select the desired cable-core and click **OK**.
It appears in the first "*Cable-core No.*" line. Proceed in the same way to assign the other cable-cores, too.

Exercise 19-3: Draw another cable with more cable-cores than available according to the assigned database type. You will receive an error message.

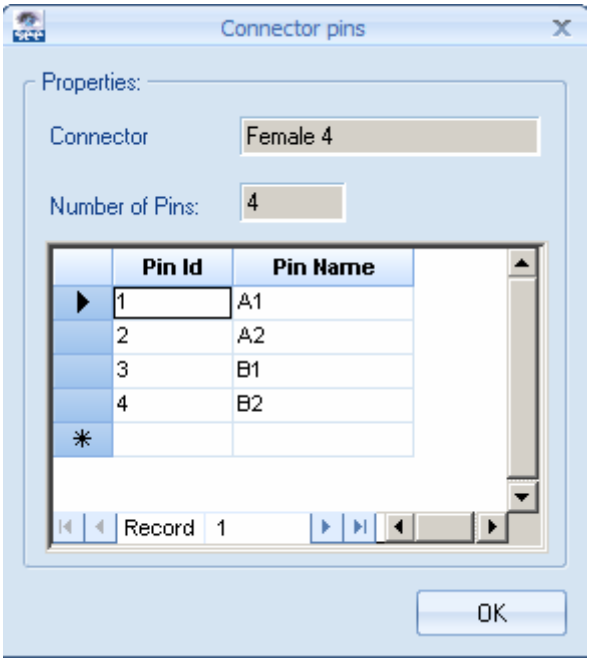
Look in the Cable editor to see which cable has too many cores.

1. Open the ***Cable*** editor.
2. Right click.
3. Select the **Check cables** pop-up command.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

U.2. MANAGEMENT OF CONNECTORS

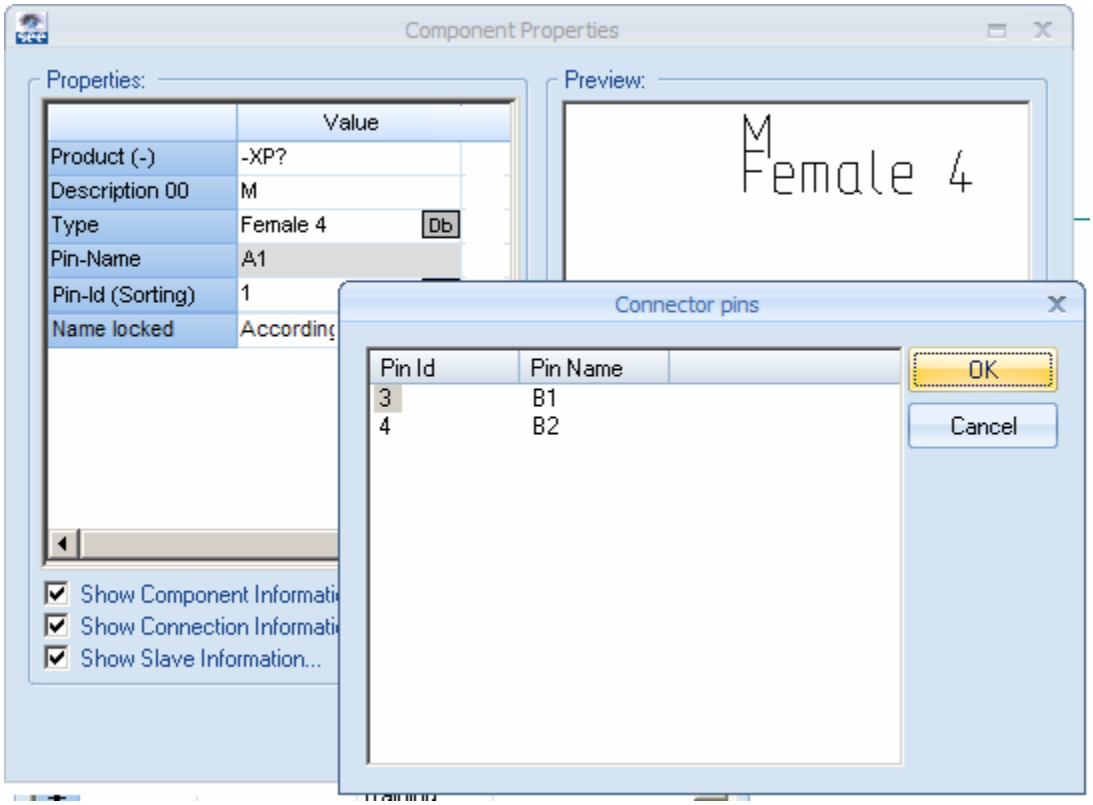
The "Pin Numbers" property allows you to define the pin numbers present in a connector.



If the pin numbers are defined correctly, *SEE Electrical* suggests automatically the pin name and controls the overbooking (if there is an overbooking, an error message appears).
The pins can get names such as 1, 2, 3... or 1A - 1Z, 1a -1z, 2A – 2Z, etc.
The "Pin Id" has to be a numerical value. The sort order of the pins have to follow the sort order of the pins in the connector.
The pins have specific sort order in the connector determined by the manufacturer or by the way the pins have to be connected to the cable.
Start with the pin in the middle and go to the ones in the outer parts.
If the Pin Id has been determined in the correct order, the pin names are suggested in the right sort order at the insertion of the connector in the circuit diagram.
When the pin numbers are defined in the type, *SEE Electrical* controls that no pin is used twice and that the overbooking is impossible.
If the type and pin information has been assigned to a connector the pin can be changed.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

- Click the Db button in the "***Pin – ID (Sorting)***" field.



The ***Connector Pins*** dialogue box displays all available pins.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

U.3. CHANNEL DEFINITION FOR PLC I/OS

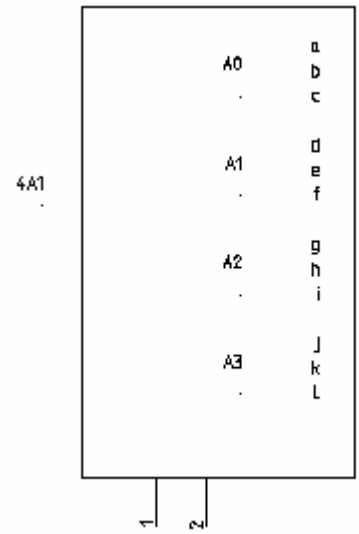
A PLC rack can be stored in the channel structure as follows:

You have a PLC rack with two external connections and four analogue I/O ports.
Port 1 has the connection points a, b, c;
Port 2 has the connection points d, e, f;
Port 3 has the connection points g, h, i;
Port 4 has the connection points j, k, l.

In this example you must add five channels to the channel definition.

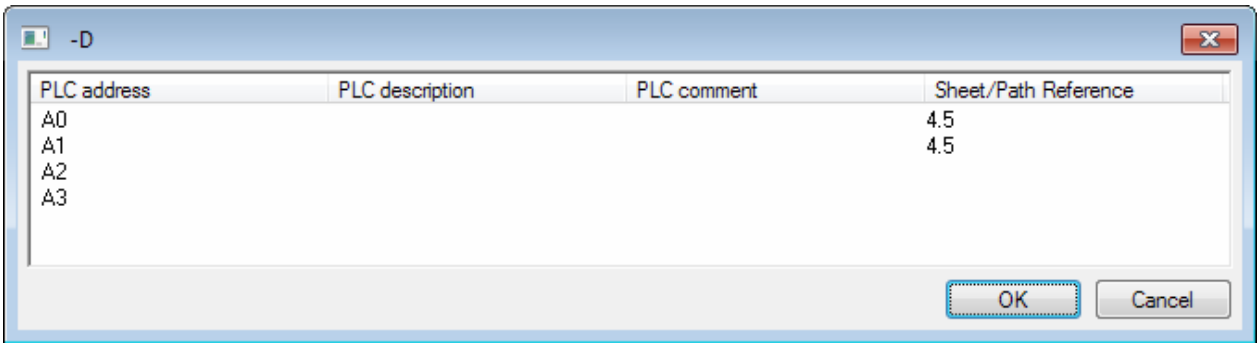
	Connection	ID
	1,2	PLC, Main
	a,b,c	PLC, Signal
	d,e,f	PLC, Signal
	g,h,i	PLC, Signal
	j,k,l	PLC, Signal
*		

If you assign this type to a PLC Rack symbol, the connection texts are taken from the channel definition.

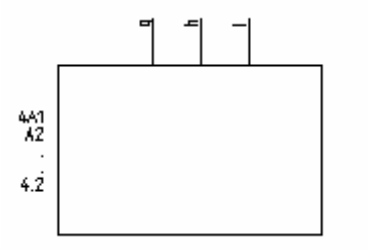


COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Then you can insert the addresses.



The connection texts are displayed.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

U.4. BLACK BOXES

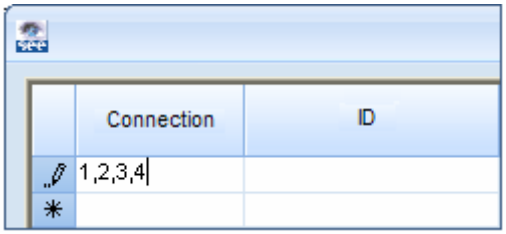
U.4.1. ASSIGNING OF TYPES TO INTELLIGENT BLACK BOX SYMBOLS

If a type is assigned to an intelligent black box, the connection texts are filled from the channel definition as usual. In this case you must draw the wires in the sort order the connection texts are defined in the channel definition to make sure the correct connection is attached to a wire.
Please choose "intelligent black box" when defining the channel information in type database.

U.4.2. CREATING A BLACK BOX FROM CHANNEL INFORMATION

Symbols for Black boxes can be generated automatically if you define the necessary values in their channel definition.

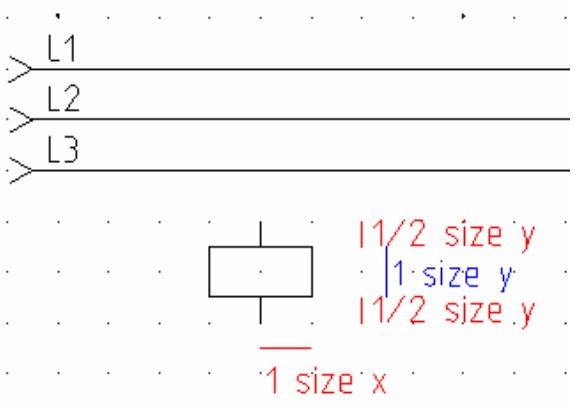
If you use this way of generating black boxes, a symbol of "component" type is generated and NOT an "intelligent black box.



Fill in the appropriate number of connection texts in the "***Connection***" field, separated by comma.

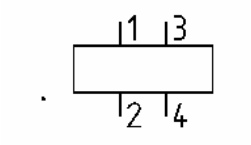
In the "***ID Circuit diagrams (EN)***" field, select the desired symbol type, for example "*Component*" or "*Coil*".

The size of the automatically created symbol depends on the default distance used for drawing potentials.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Position the newly created type via one of the commands **Functions □ Component □ Add**, **Functions □ Other □ Pick list** or by using the ***Component Explorer***. A black box is automatically created.
The example above generates the following result:



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

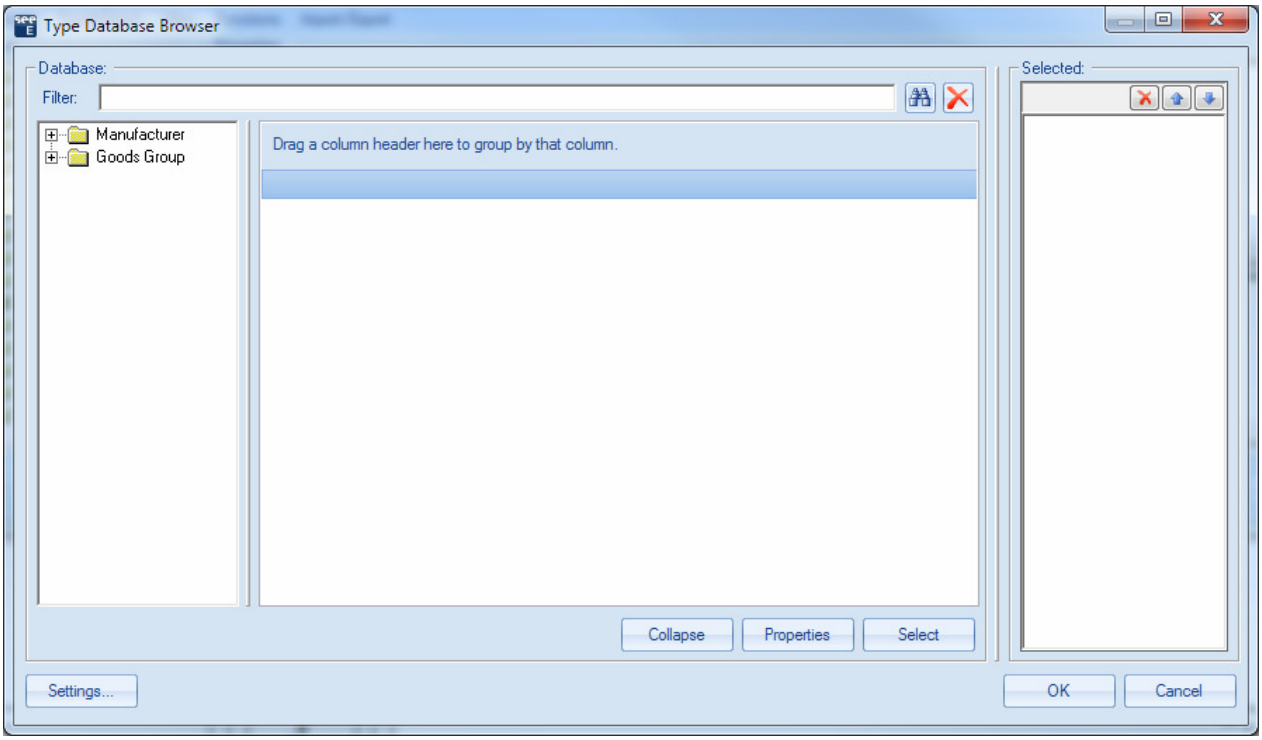
V ADDITIONAL HINTS ABOUT THE TYPE DATABASE

(Standard)

V.1. SEARCHING AND COPYING IN THE TYPE DATABASE

✓ Searching

In the ***Type Database Manager*** window, the **Functions** □ **Find Type** command is available. It opens the familiar ***Type Database Browser*** dialogue, allowing you to search for a type.



✓ Copying

In the ***Type Database Manager*** window, the **Functions** □ **Copy Selected Type** command is available. By generating a copy of the selected type, this command allows you to create new types quickly by modifying existing types.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

V.2. COMPLETING COMPONENTS

If a component consists of several parts, you can use the **Functions** □ **Component** □ **Complete** command or activate the icon to insert the parts that are not inserted yet.

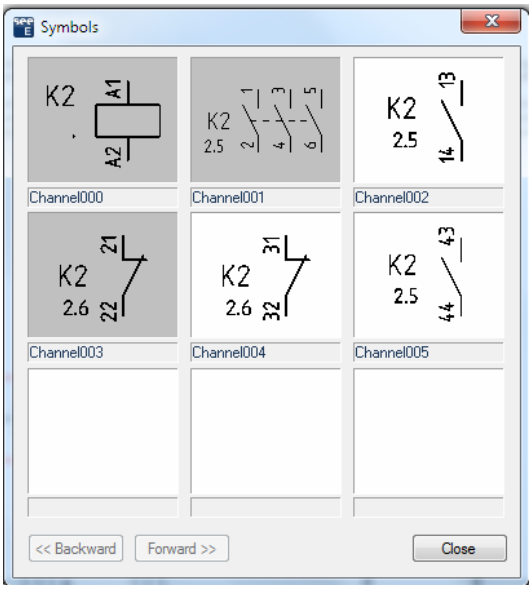
Typical components consisting of several parts are:

Contactors: They consist of a coil and several contacts.
Some motor protective switches: They consist of the component itself and of contacts not located at the same place as the component.
PLC-Racks: The Rack itself and several signals may be used.

Requirements:

In order to use this function, it is necessary to define in the Channel definition of the component and all its parts which symbol must be used in the *Circuit diagram* module.

This applies to all components in the workspace.
At least one part of the component must already be inserted in the drawing.



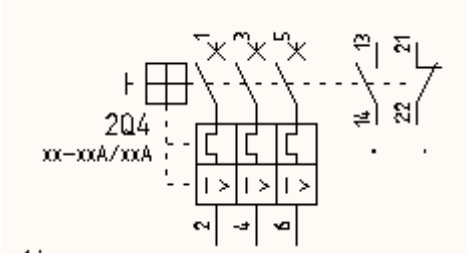
Channel definition:
In the Type Database, the Circuit diagram symbol must be specified in the Channel definition.

Connection	ID	Symbol	Reference Symbol
A1,A2	Relay Coil	EN61346-2UK\Relay coils\1-pole Db	Db
13,14	Relay-contacts, NO	EN61346-2UK\Relay-contacts, NO\1-pole NO Db	Types\Mirrors\NO Db
21,22	Relay-contacts, NC	EN61346-2UK\Relay-contacts, NC\1-pole NC Db	Types\Mirrors\nc Db
33,34	Relay-contacts, NO	EN61346-2UK\Relay-contacts, NO\1-pole NO Db	Types\Mirrors\NO Db
43,44	Relay-contacts, NO	EN61346-2UK\Relay-contacts, NO\1-pole NO Db	Types\Mirrors\NO Db

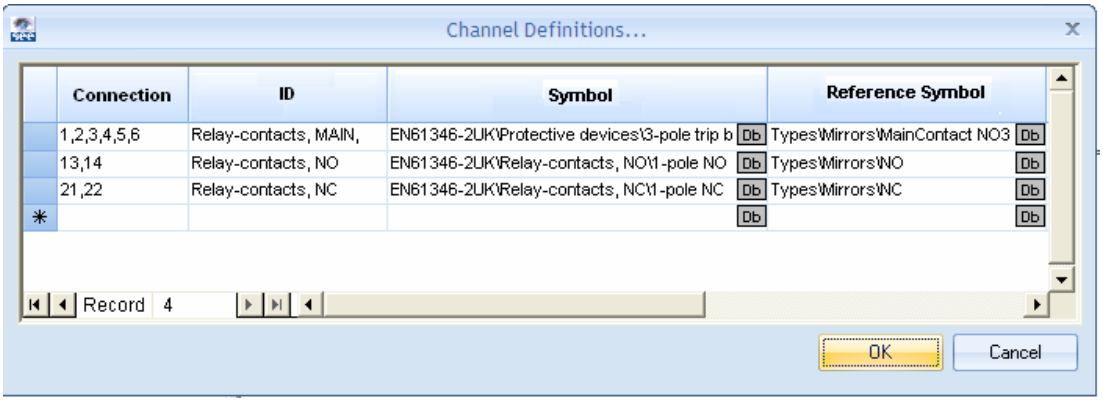
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Components with auxiliary contacts/contactors/relays:
A component with auxiliary contacts (contactors/relay) doesn't consist only of one symbol in the Circuit diagram but dependent contacts (Slaves) are placed at other places different from the place where the main component (the Master) is located.

The motor protective switch below has 6 connections (1, .., 6) directly connected at the component and two dependent contacts (NO and NC) located and connected at another place in the Circuit diagram. There is only a cross-reference to the Slaves at the Master component and vice versa.



Therefore, you will find 3 lines in the channel definition:
A line for the main component (with connections 1,2,3,4,5,6)
A line for the NO contact (with connections 13,14)
A line for the NC contact (with connections 21,22).

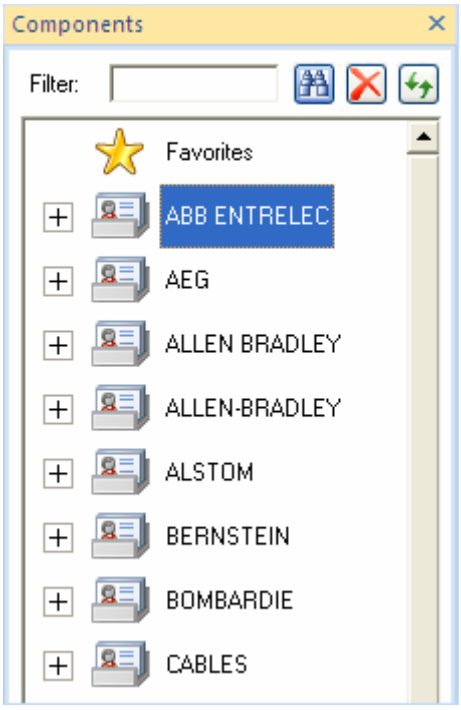


The channel definition for a contactor/relay is formed in a similar way.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

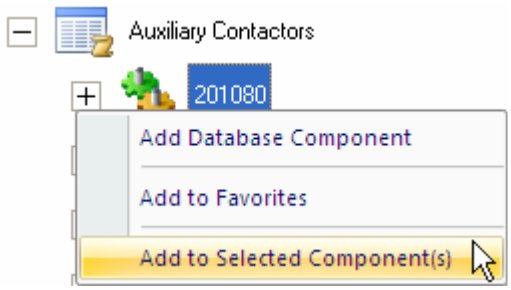
V.3. USING THE COMPONENT EXPLORER

The ***Component Explorer*** allows you to handle components with type information.
To display the ***Component Explorer***, select the **Home** □ **View** □ **Components** command.



By using this explorer, you can:

✓ Easily attach types to your components, as well as to multiple components in one step.
To do this, add one or several components, find the desired in the ***Component Explorer*** and right-click to activate the **Add to Selected Component** pop-up command.

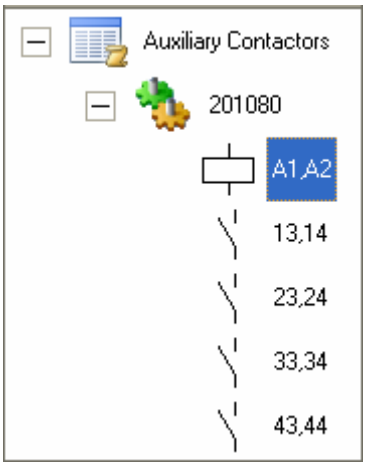


✓ Use the Favorites folder for types often used
The handling of this Favorites folder is the same as the one in the Symbols Explorer.

✓ Use the **Add Database Component** pop-up command
If the channel information is properly defined, the **Add Database Component** pop-up command allows you to add the selected type into the drawing, as already known via the **Functions** □ **Component** □ **Add** command.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

✓ Insert symbols directly from the ***Component Explorer***
If the channel definition is correct, the symbols to be used in the Circuit diagram module are visible in the Component Explorer and can be "dragged" into the drawing.



Note

If you add a new type in the Type Database, you need to refresh the Component Explorer view to be able to access the new type in the Component Explorer. To refresh the view click the icon.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

V.4. IMPORTING AND EXPORTING ARTICLE MASTER DATA

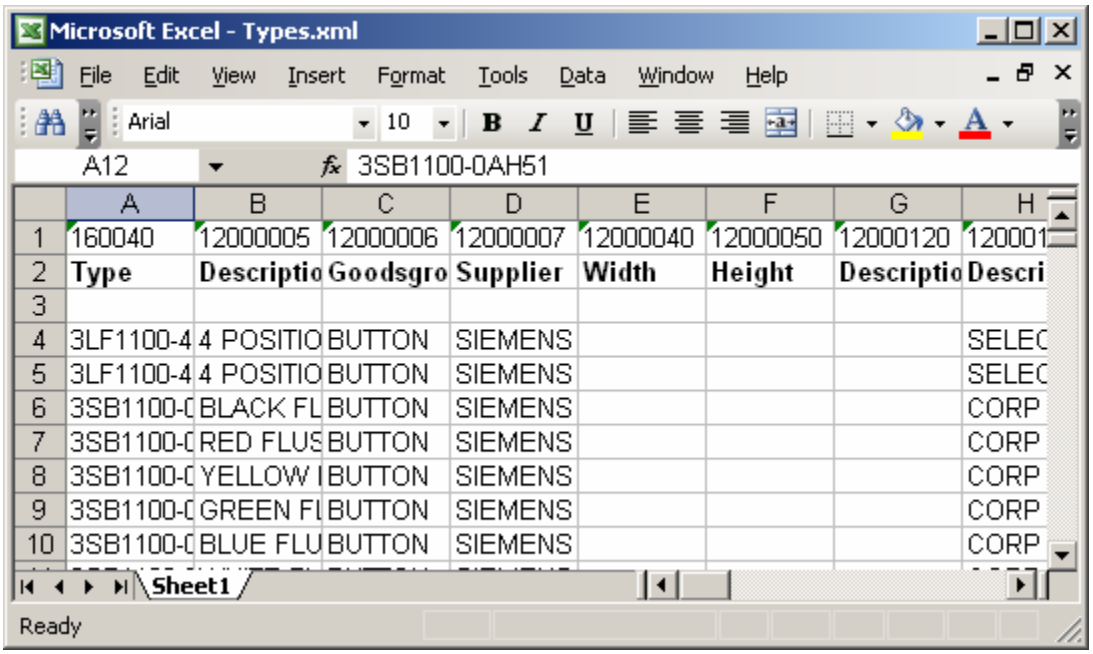
You can receive article master data from the component manufacturer or from another source and use it. *SEE Electrical* offers the possibility to import data with *ECAD* or *Excel* format in the type database.

You can find both functions in the ***Type Database Manager*** window, in the **Import/Export** menu.

Here, you are also given the opportunity to import types from a *XML* spreadsheet file, for example created with a *Microsoft Excel* application. However, a *XML* file may be generated from many other external programs. As well, exporting types to a XML spreadsheet is possible.

The *XML* spreadsheet represents an *ASCII* formatted text file presenting the data structured in rows and columns in a standard file structure, predefined with tags.

It is possible to open/edit/save *XML* Spreadsheets using a *Microsoft Excel* application.
The .xml file looks very much like a standard *Excel* table, but it cantinas hidden control elements.
That is why you can change only what is described below.



	A	B	C	D	E	F	G	H
1	160040	12000005	12000006	12000007	12000040	12000050	12000120	120001
2	Type	Descriptio	Goodsgro	Supplier	Width	Height	Descriptio	Descri
3								
4	3LF1100-4	4 POSITIO	BUTTON	SIEMENS				SELEC
5	3LF1100-4	4 POSITIO	BUTTON	SIEMENS				SELEC
6	3SB1100-0	BLACK FL	BUTTON	SIEMENS				CORP
7	3SB1100-0	RED FLUS	BUTTON	SIEMENS				CORP
8	3SB1100-0	YELLOW I	BUTTON	SIEMENS				CORP
9	3SB1100-0	GREEN FI	BUTTON	SIEMENS				CORP
10	3SB1100-0	BLUE FLU	BUTTON	SIEMENS				CORP

- ✓ The first line contains the ID from *SEE Electrical* for each column. You are not allowed to change this line.
- ✓ The second line contains the titles of the columns. You are not allowed to change this line.
- ✓ The following columns contain the information about the article in *SEE Electrical*.

Save the *Excel* file. Rename it, if desired, and keep in mind that you have to select the file type.

The contents and the structure of the *Excel* file can change, depending on the information contained in the exported *SEE Electrical* type database.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

V.5. MODIFYING EXISTING TYPES IN THE PROJECT

It is possible that you may need to make changes to a type that you have already assigned to a symbol in your project.

These changes will not affect directly the current project, because the types used in the project are stored and cannot change automatically when the type database is changed.
For example, if you negotiated prices for your project a year ago, the information cannot be updated automatically.
To update the current type information, execute the **Functions** □ **Types** □ **Update** command.

V.6. DISPLAYING INFORMATION ON THE EQUIPMENT FROM THE TYPE DATABASE

You can display information from the type database on a piece of equipment inserted in the circuit diagram. In order to do this, you must use texts with the respective attributes during the symbol generation.
The necessary text attributes can be found in the *Type Properties* node in the ***Text*** dialogue.

V.7. DEFINE WHICH TYPE DATABASE TO USE WITH A PROJECT

(Standard and Advanced)

It is possible to use different type databases in *SEE Electrical*.
Because of this each company can name its type database according to their needs - *<MyCompanyTypes.SES>*.
If the type database carries the companies name, it might be much more convenient to include this file into the process of securing data.
Customers serving different clients of course will also benefit from this as they now can assign the clients type database to the client workspace(s).
If a type database different from the "*TYPES.SES*" is used, this is defined via a command. The **Set type database** command allows you to select the new database to use with the current workspace.

The definition is saved in the workspace (or workspace template).

Assign type database to multiple workspaces

The **Set MultiType DB** command allows to assign a type database to multiple workspaces or to all in one folder. It can only be executed, if no workspace is open.

First select the type database to use via the "***Select type database***" field.

The icon allows you to select several workspaces.

The icon allows you to select folders, where all workspaces will be treated.
"***Set type database***" applies the type data base selected to all workspaces defined.

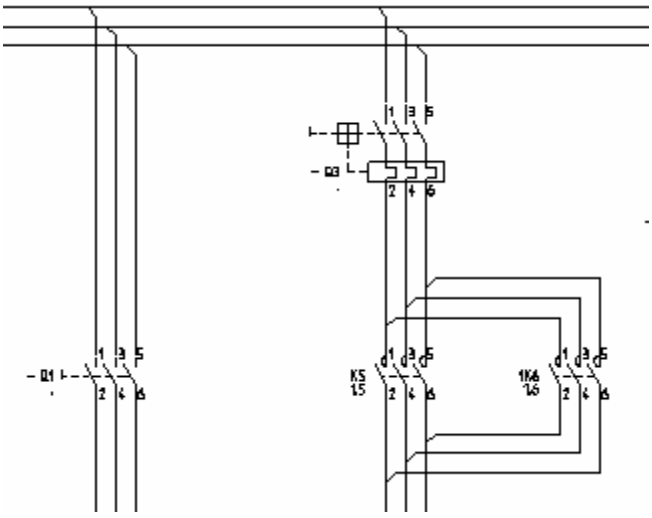
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

W WIRE PROPERTIES

(Standard)

W.1. WIRE DIRECTION

SEE Electrical standard offers you the possibility to use wire direction instead of the node.



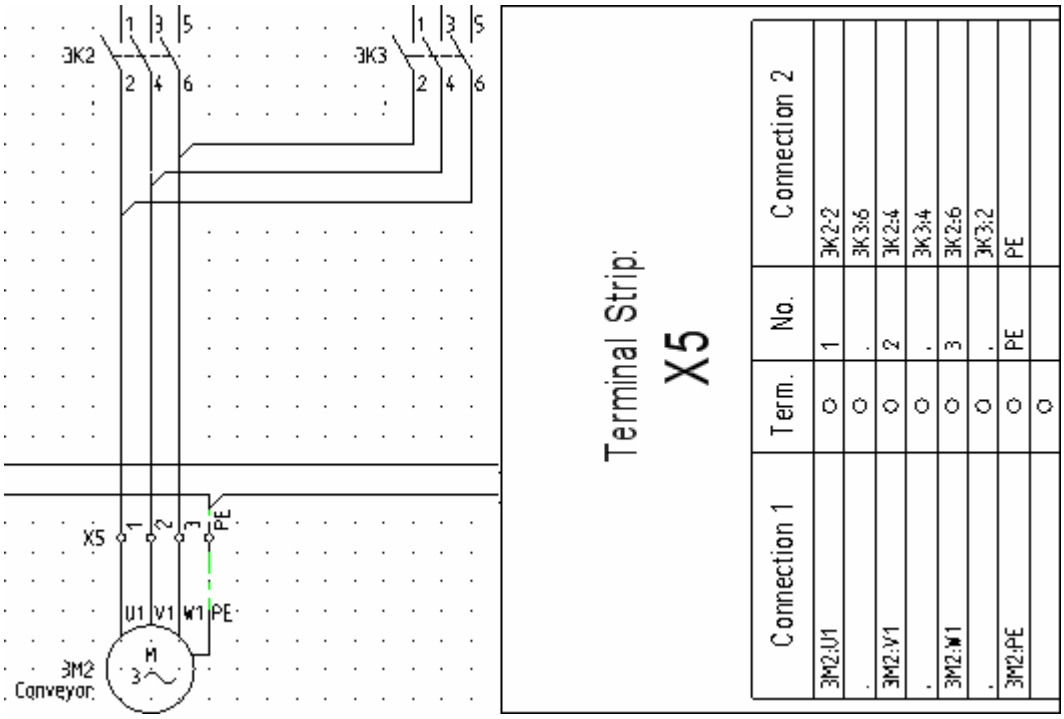
You can change the wire direction via the **Electrical** □ **Wires** □ **Direction** command.

- Click on the node, the wire direction will be changed.
- Right-click *Circuit Diagrams* within the ***Workspace Explorer***.
- Execute the **Properties** pop-up command and click the ***Wires*** tab.
- Tick the "***Show Wire directions***" option to display wire directions permanently.

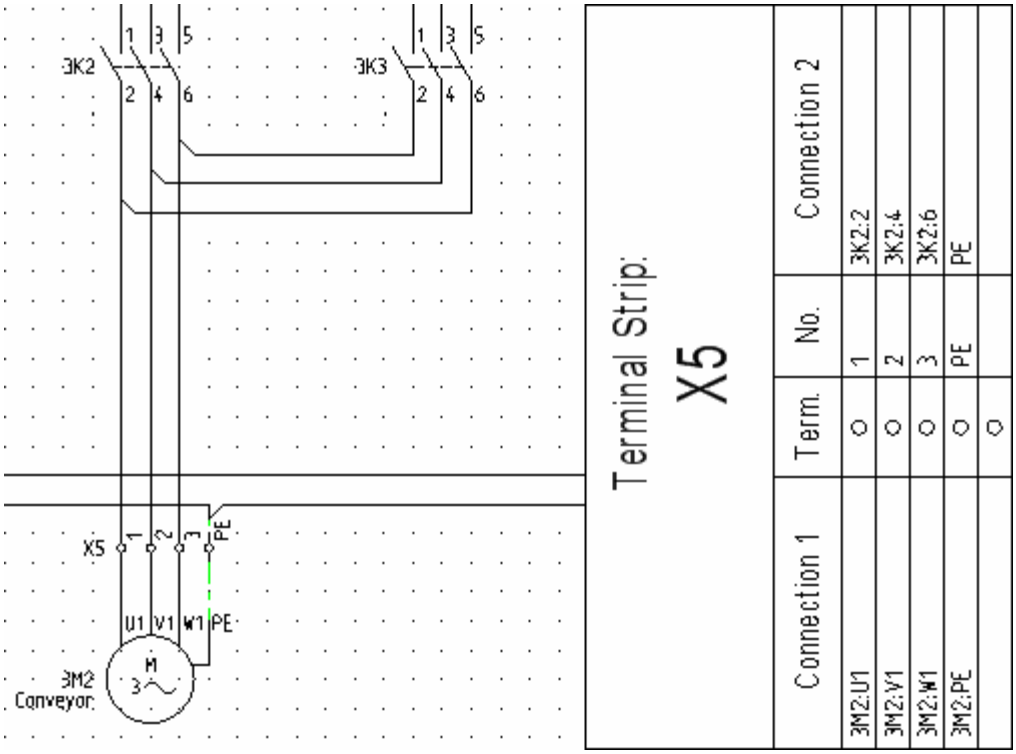
By using wire directions, you can change the appearance of the terminal matrix and in the *advanced* level, the terminal plan as well.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Example 1:



Example 2:



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Note :

You can change the default wire directions via the following registry key: [HKEY_CURRENT_USER\Software\CAE Development\SEE Electrical\Version V7R2\1000\WireDirections] or [HKEY_CURRENT_USER\Software\CAE Development\SEE Electrical\Version V7R2\1001\WireDirections] for circuit diagrams IEEE.

The possible values are:
WireDirectionLeftRightDown: 15025, 15027, 15035 and 15037 (default EN)
WireDirectionLeftRightUp: 15026, 15028, 15036 and 15038 ((default EN))
WireDirectionLeftUpDown: 15022 ((default EN)), 15024, 15032 and 15034
WireDirectionRightUpDown: 15021 ((default EN)), 15023, 15031 and 15033
You have to be an administrator on the computer to be able to perform this operation.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

W.2. CHANGING TARGETS

In the terminal matrix, internal and external targets must appear on one defined side.

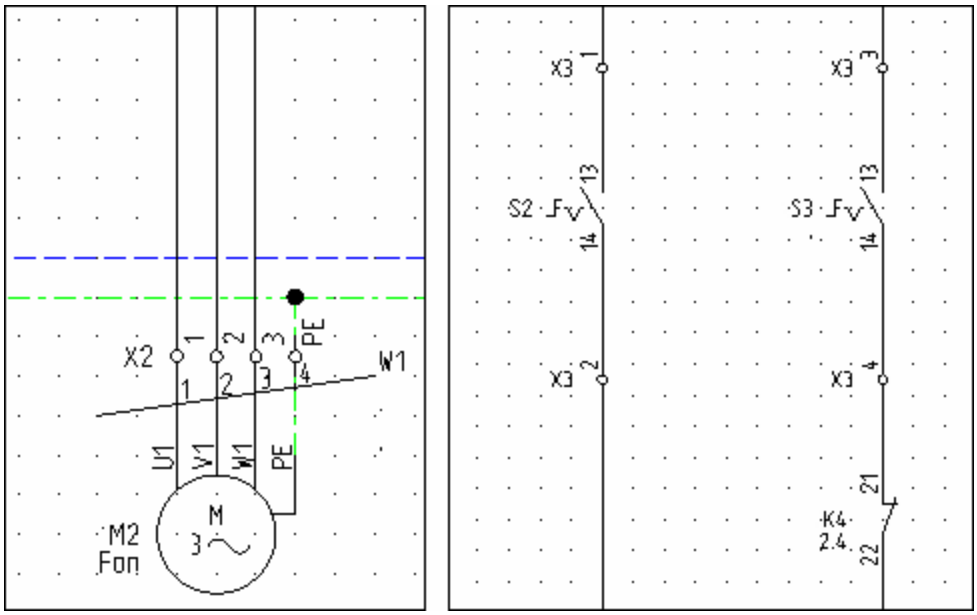
Terminal Strip:

X3

Connection 1	Term.	No.	Connection 2
2S2:13		1	1Q2:14
2K2		2	2S2:14
2S3:13		3	1Q4:14
2K4:21		4	2S3:14
2S4:13		5	.
2K3:21		6	2S4:14
2P6		7	2K2:14
N		8	2P6
2P7		9	2K3:14
.		.	2K4:14
N		10	2P7

In the example above, internal and external targets are shown in colour either on the right or on the left side.
This comes from the circuit diagram as terminals find usually the internal targets at their upper connection, and the external targets at their bottom connection.

This can be useful in some cases but in others - not:



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

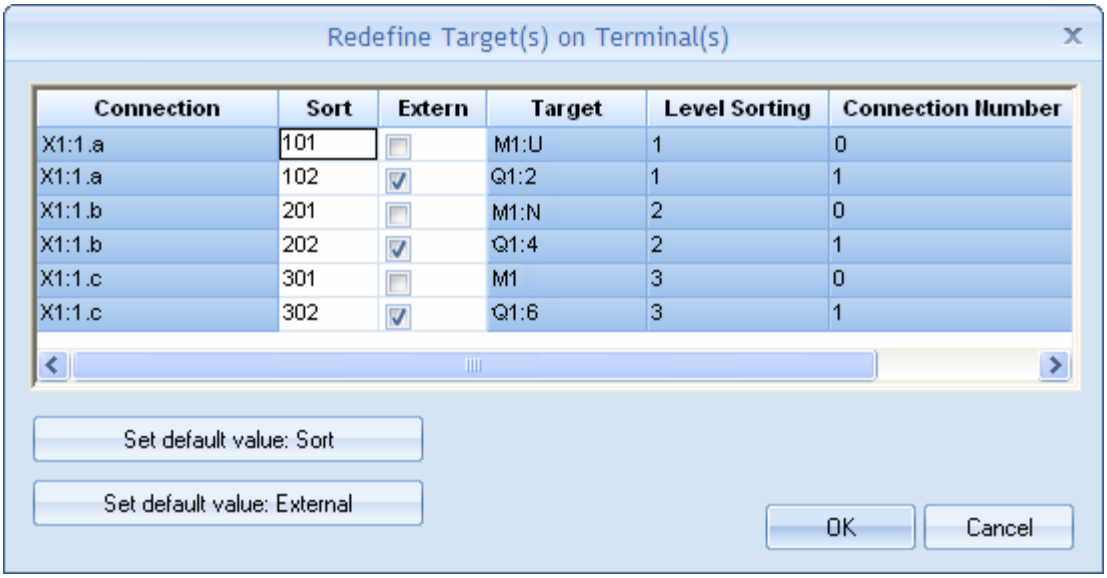
Exercise 21-1: Swap the internal and external target for both bottom terminals in the example displayed above on the right.

1..CA **Electrical**
2..CO **Connection** (**View** panel)
Each first connection within a component is marked in yellow.
The first connection of a terminal manages the internal target.
3. Select the terminals whose connections you wish to swap.
4..CA **Electrical**
5..CO **Swap** (**Connections** panel)
The connections of the highlighted terminals have been swapped.

Exercise 21-2: For terminals with more than 2 connections or Multi-Layer terminals, you can define more external/internal targets.

1. Select the terminals whose targets you wish to redefine.
2.CA **Electrical**
3.CO **Redefine target(s) on terminal(s)** (**Connections** panel)

The following dialogue appears:



4.> Use the check box within the "***Extern***" column to set the connections as external or not.
5.> Click **OK** to validate.
The connections of the selected terminals are redefined.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

W.3. DEFINING A LINK

Links are indicated automatically from the *SEE Electrical standard* level. Two types of Links are created: Links identified via potentials and links identified via wires.
You can assign another type to the links, as well as the standard type. In this way you can differentiate between inlaying links, wire links etc.

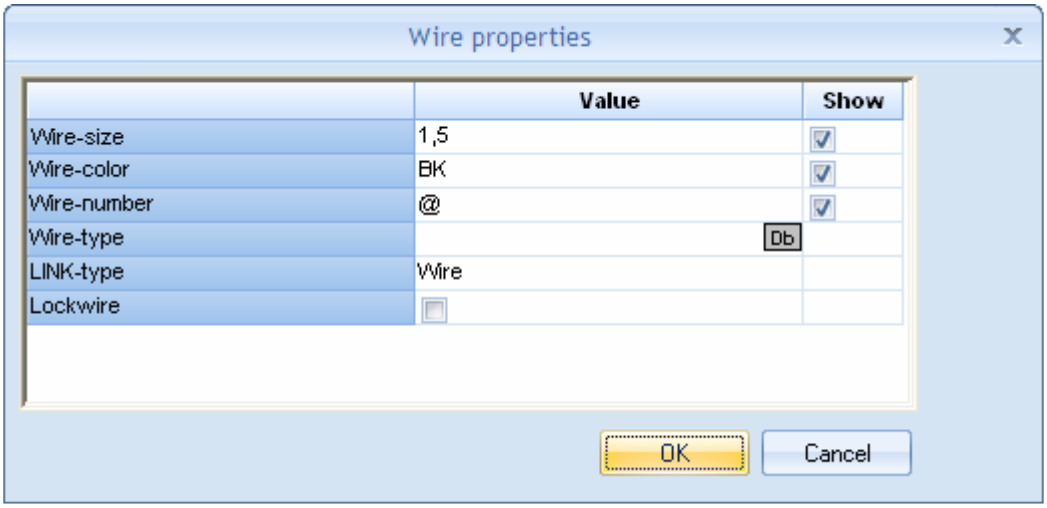
In the terminal matrix, the different link types can be displayed on various locations.

The following definitions are used:

P	Links identified via potentials
0	Links identified automatically via wires (main link type)
1	Link type 1, you can assign it instead of the Wire type, for example inlaying link
2	Link type 2
etc. up to	
10	Link type 10

Exercise 21-3: Switch to page 2 of your training workspace. Place a new terminal X2:11 beneath the terminal X2:1. Define the link type between the terminals X2:1 and X2:11.

1..CA **Electrical**
2..CO **Properties** (**Wires** panel)
3.+ Identify the wire whose LINK-type you want to change.

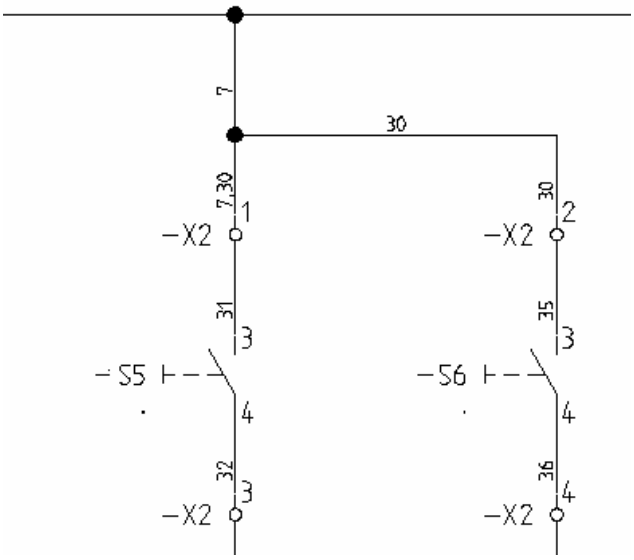


4.> LINK-Type
The value "*Wire*" in the **LINK-type** line means that a link type is not defined. In the **training,** the LINK type "*Wire*" must represent inlaying links. Bridge type **1** would represent wire links.
5.> Bridge type 01
Select another link type.
6.> **OK**
The link type has been changed and you can identify the next wire or close the function.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

W.4. GRAPHICAL WIRE NUMBERING

The graphical wire numbering is frequently requested. *SEE Electrical standard* provides this feature.

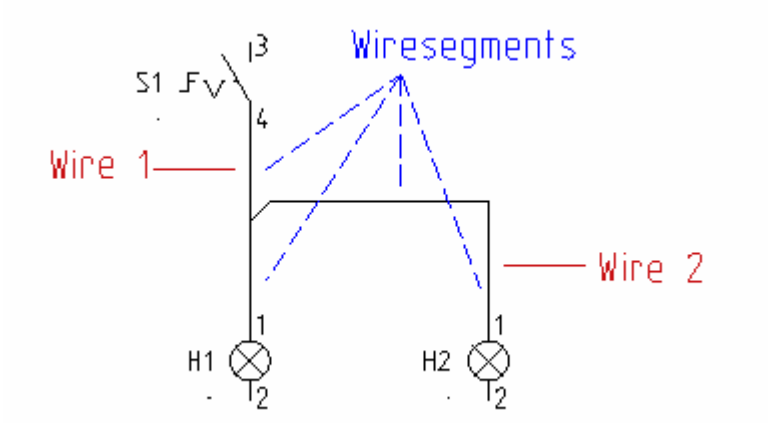


The functions for automatic numbering and removing wire numbers are available in the **Electrical** □ **Wires** □ **Numbers** subpanel.

It is important that the terms "net", "wire", "wire segment" are clearly defined in case signal types of wires are not used. In case you use signal types of wires you have to understand well the terms "signal type", "net", "wire", and "wire segment".

Signal type

Signal types can be: Power, Control, N, PE, Data bus, etc. For each signal type you can define attributes (colour, size, line width and visibility for wire texts).
You will find detailed information in the next chapter.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Net
In the example shown above is present one net that links the switch with the two lamps. A net can connect different connection points from different components. A net consists of different wires.
If you use the signal type, different nets can have the same signal type.

Wire
The wire is the physical object that always links exactly two components. The net shown in the example above consists of two physical wires.
- One wire is connecting the S1 switch with the H1 lamp
- One wire is connecting the H1 lamp with the H2 lamp.

The wire directions in the net are very important.

Wire segment
Wire 1 contains two segments and Wire 2 has three segments. The segment between lamp H1 and the direction node is common for both wires.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

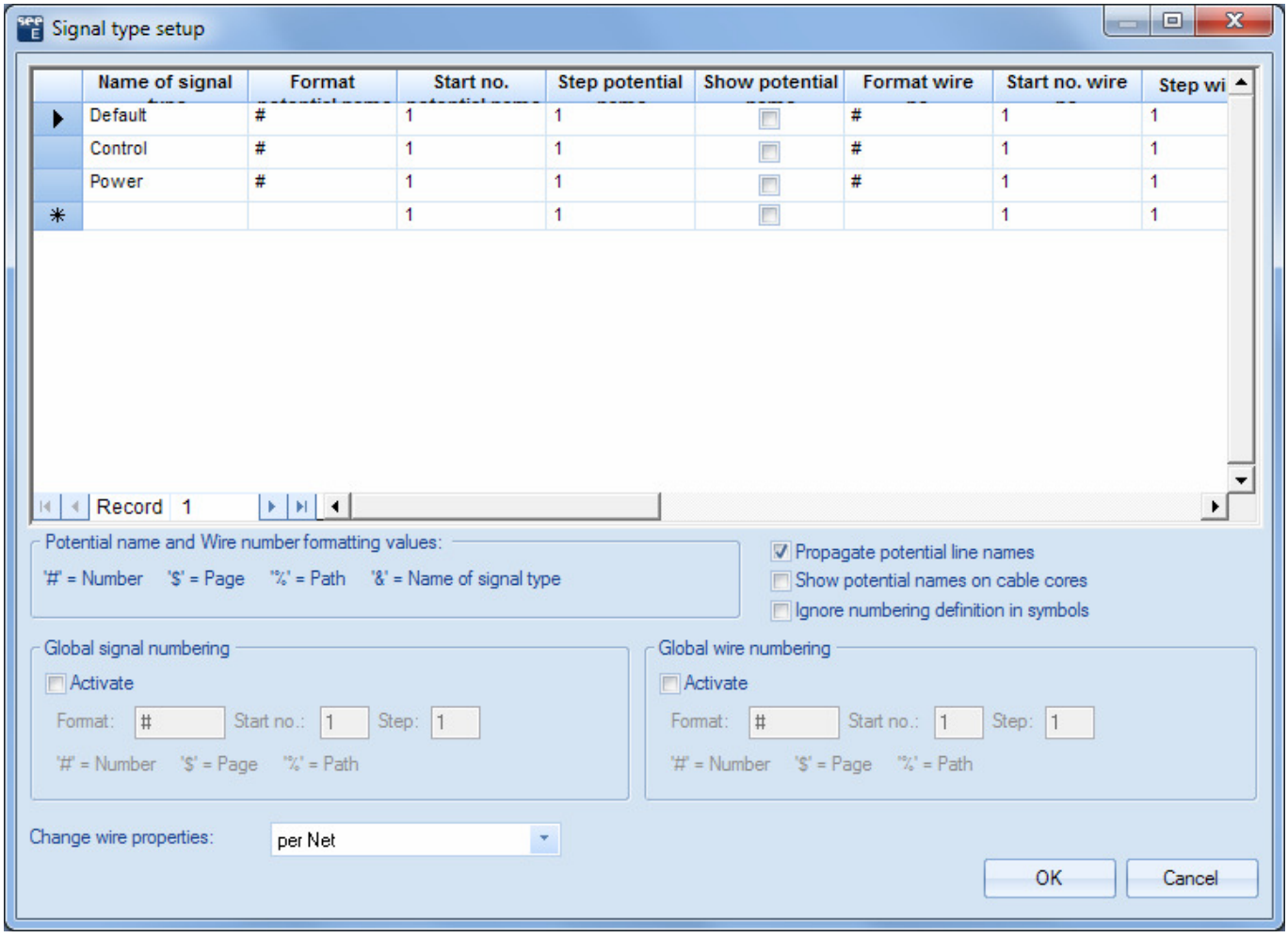
W.5. SIGNAL PROPERTIES

Signal properties

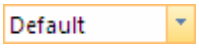
It is possible to assign *Signal* properties when drawing wires - for example, for control circuit or for power circuit. For each Signal type you can define different attributes such as wire colour or size.

First, it is necessary to activate the "***Signal types for wires***" option in the ***Wires*** tab of the ***Circuit Diagram Properties*** window, available after executing the **Properties** pop-up command for Circuit diagrams in the ***Workspace Explorer***.

Afterwards, click the **Signal type setup** button to define the desired signal properties. For the *standard* level, only 4 Signal properties are possible.



Different options are available here. For more information, refer to the software's help.

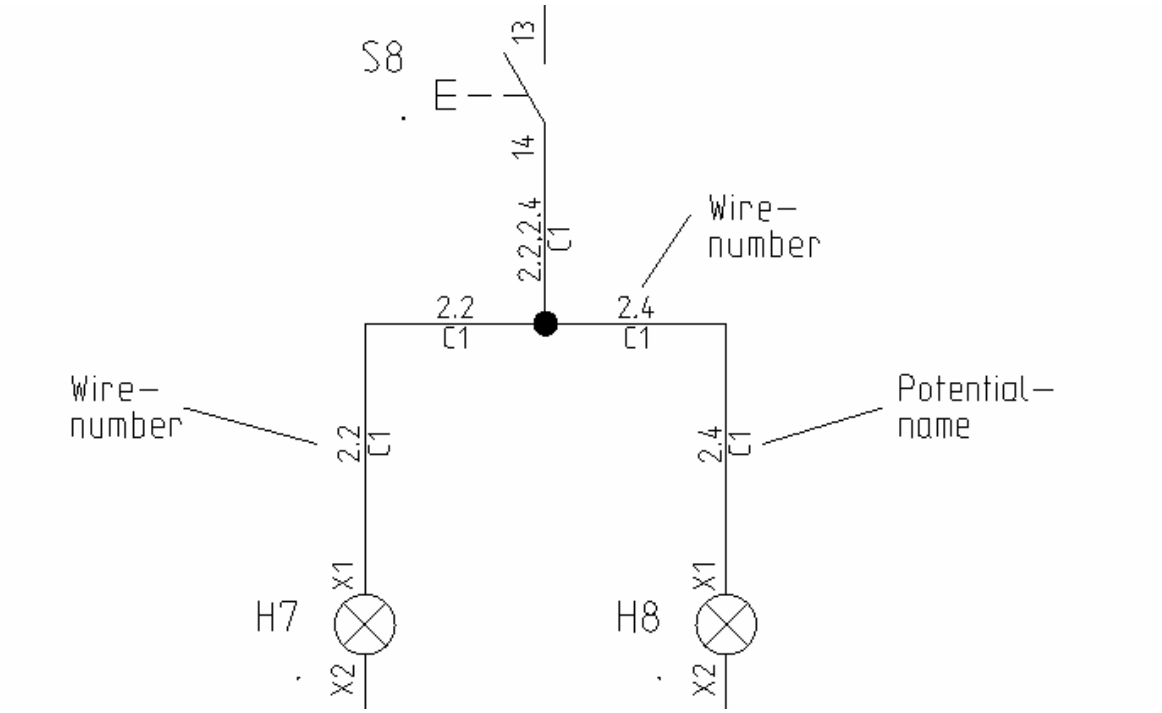
If you work with signal properties, you can select the signal property from the 

menu in the **Electrical** category before you start drawing your potentials.

The properties, as you have specified them for the corresponding signal type, are displayed subsequently in the drawing.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Example:



You can use various ways to assign the desired values to the wires. It is your choice which one to use.

W.6. WORKING WITHOUT SIGNAL PROPERTIES

If you do not want to use signal properties, you can generate wire numbers and define the colour and size for your wires in case you need these values for your wire list.
The "***Potential***" and "***Unique***" options in the ***Wires*** tab of the ***Circuit Diagrams Properties*** window define how the attributes are managed.

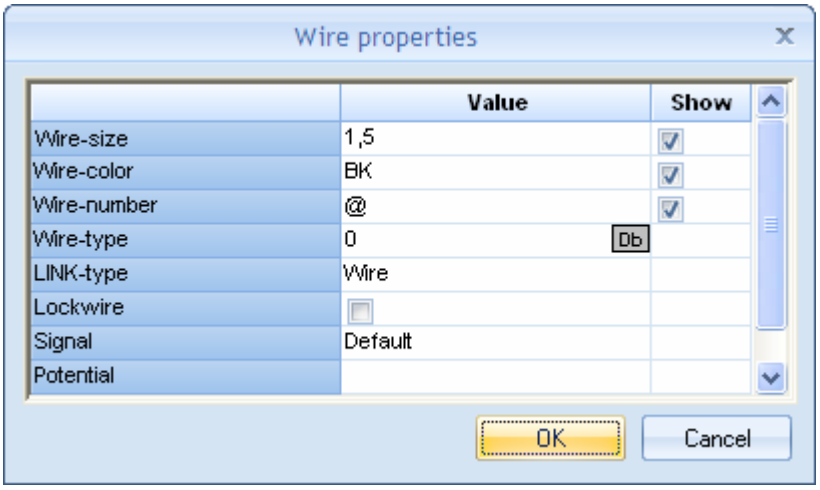
- Potential – all wires in one net have the same attributes such as colour, size and wire number.
- Unique – each wire can have its own attributes.

You can use various ways to assign the desired values to the wires. It is your choice which one to use.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

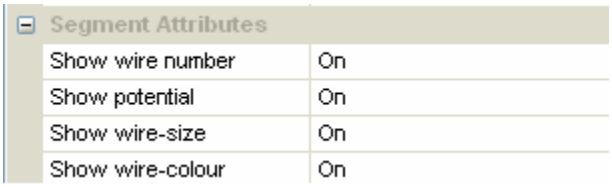
W.7. VIEW AND CHANGE WIRE ATTRIBUTES

You can control the wire attributes defined for a specific wire. The following dialogue appear when you double click a wire:

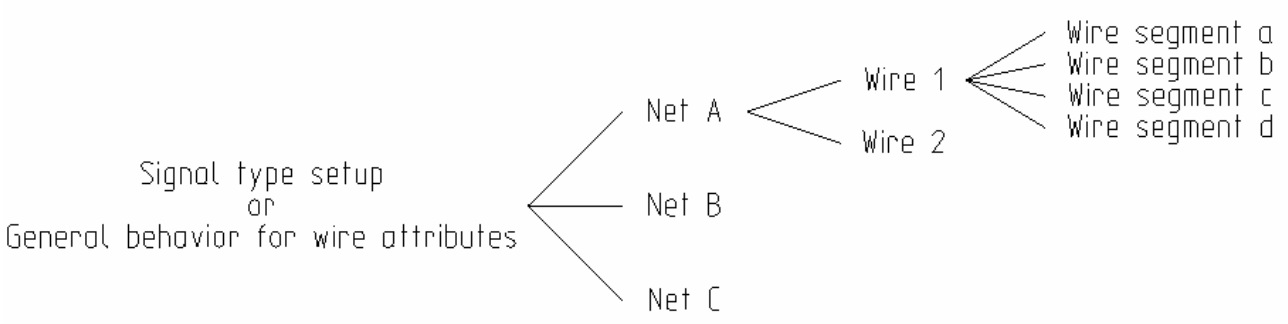


The information about the colour and the size of the wire is important if you work with Wires list. You can use various ways to assign the desired values to the wires. It is your choice which one to use.

In the ***Properties*** window of a wire you can switch on/off the visibility for the texts of the wire segments.



The segment text will be displayed only in case a wire text is displayed from the ***Circuit diagram Properties*** and the ***Wire properties***. If the visibility is switched off for a wire, the option for the wire segment will not have any importance. The visibility is determined in the following way:



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

W.8. DEFINE THE POSITION OF THE WIRE TEXT

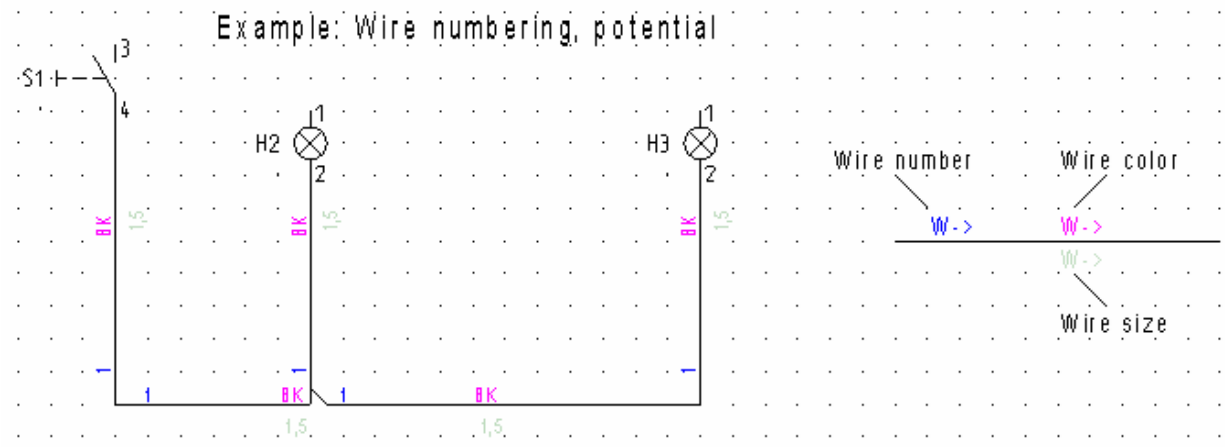
It is possible to define the position of the text attributes for the wire number, colour and size. If you use Signal type of wire, it is possible to define additionally the attributes and the position of the potential name.
The positions of the wire properties have to be defined in a symbol. To define the settings for the horizontal and vertical wires you need to create two symbols – one for the horizontal wire and one for the vertical wire.
Each symbol has to contain one line which represents the wire and the texts for the wire number, the potential name, wire size and colour (the text have to have the text attributes (ID) for wire number, potential name, wire size and colour. The text properties such as text font, width and height, colour and angle will be used when you create the wire properties.

If there is no text representation in the symbol for one of the wire properties, the default settings will be used if the property will be displayed.

The position of the wire property on the wire will be taken from the position of the corresponding text in the symbol. The value of the text defines how will be treated the position in relation to the wire.
The following values are possible:

"**W->**" the position of the text will be calculated from the left extremity of the wire.

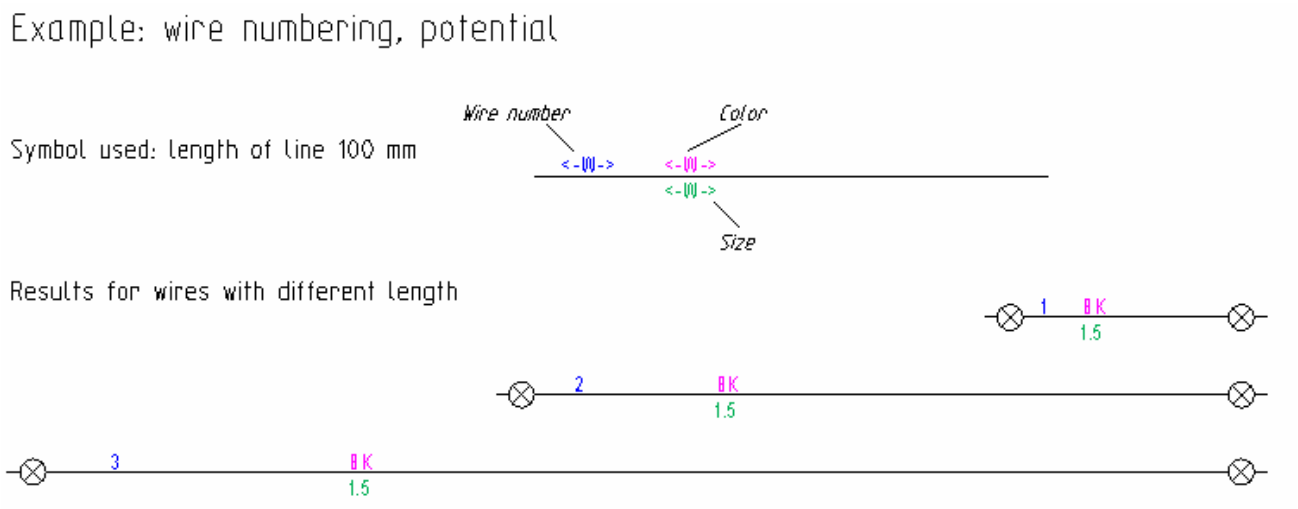
For example: the position of the text (dX, dY) in relation to the extremity of the line in the symbol is 5 mm. The "W->" value is defined in the text with the ID for the wire number.
When the wire number is created in the wire, it will be places at 5 mm from the extremity of the wire regardless of the length of the wire.



"<-W" the position of the text will be calculated from the right extremity of the wire. The alignment will be from the right extremity of the wire.

"<-W->" the position of the text will be calculated with the percent between the position of the wire property in relation with the wire.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



The symbol for the horizontal wire is supporting Movimenteble x- coordinate and the symbol for the vertical wire is working with Movimenteble y-coordinate.

Important: After you change a symbol, you have to restart SEE Electrical.

The symbols have to be defined in the registry of Windows in the following way:
<symbol database>\<symbol folder>\<symbol name>

The registry value for the horizontal wire symbol is ...\1000\WirePropSettings.
The registry value for the vertical wire symbol is ...\1000\WirePropSettingsV.
The setting is used for Circuit Diagrams EN and IEEE.
This way of defining the symbol properties is valid for all wire modes.

The registry settings
...\Settings\Text\DefaultWirePropertiesFont
and
...\Settings\Text\DefaultPotentialNameFont

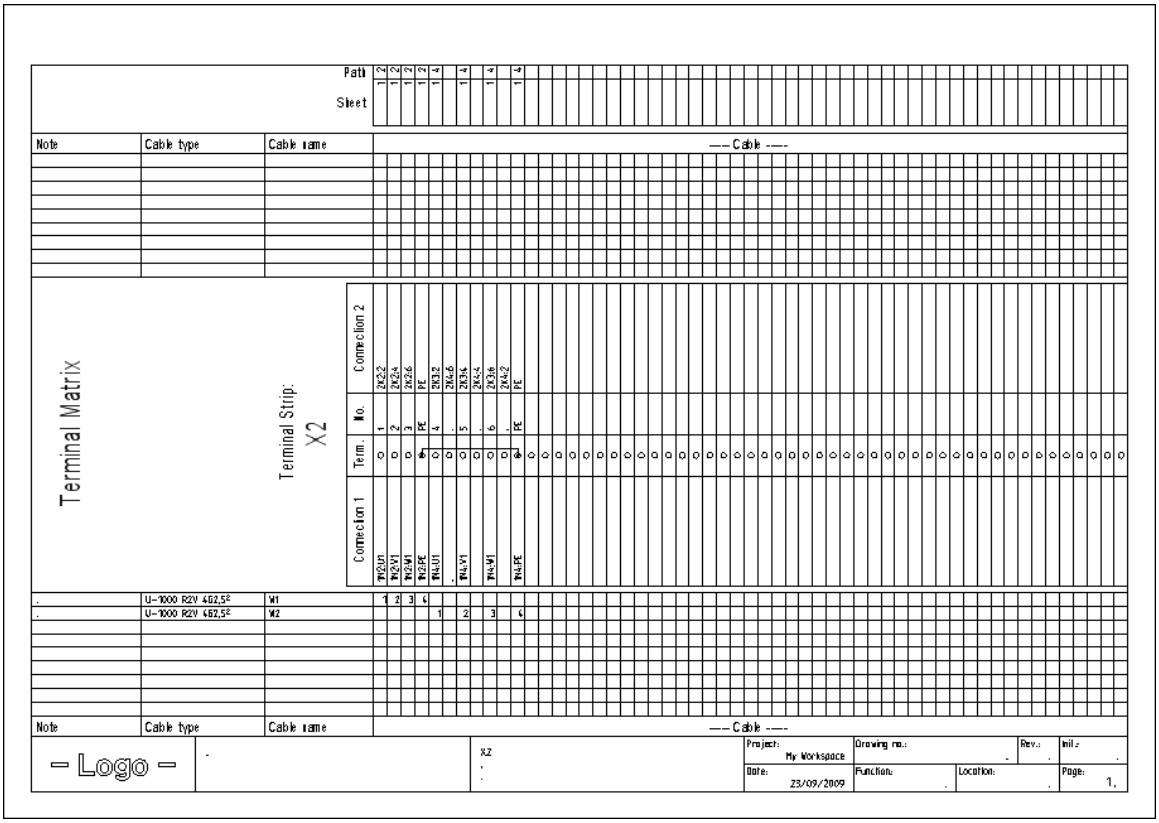
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

X TERMINAL MATRIX

(standard)

X.1. GENERATING A TERMINAL MATRIX

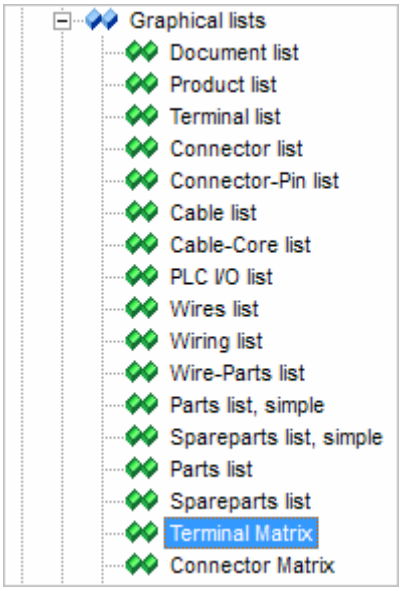
The terminal matrix facilitates the installation of terminal strips.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercise 22-1: Generate a terminal matrix for a terminal strip from the training project.

1. Select *Terminal matrix* from within the *Graphical Lists* in the Workspace tree.
Expand the *Graphical Lists*.



2. Right-click with the mouse.
3.CO **Generate**
In the ***Select Terminal Row*** dialogue, choose the terminal strip(s) for which you wish to generate a matrix, for example X2 and X3.
4.> X2, X3
5.> OK
The Terminal matrices for the selected terminal strips have been created.

Exercise 22-2: Look at the terminal matrix. Each terminal strip appears on its own page.

1. 0001
Double-click the page 1 beneath **Terminal Matrix** in the Workspace Explorer.
2. 0002
Double-click the page 2 beneath **Terminal Matrix** in the Workspace Explorer.

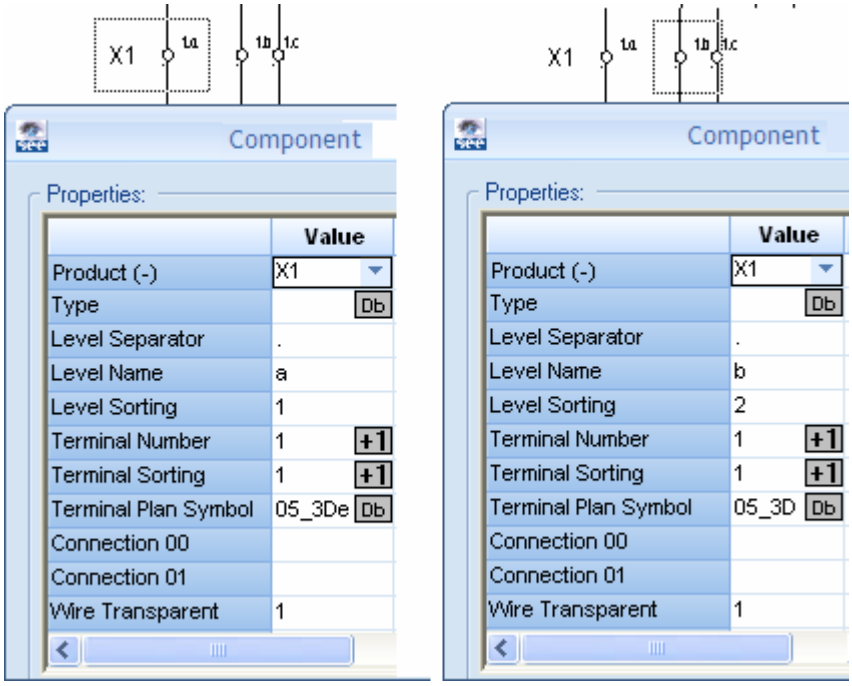
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

X.2. HANDLING MULTI-LAYER TERMINALS

You can handle multi-layer terminals in the *Circuit diagram* as follows:

- Place the multi-layer terminals.
- Enter the needed texts.

Single levels are connected together when the terminal name, terminal number, and terminal sorting are equal. (The terminal number can be also empty.)



- Select the answer "***Yes***" to the question if the components are to be combined.

In the terminal matrix, the terminal number, the level separator and the level name are entered together in the text placeholder for the terminal number.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

X.3. CREATING A TEMPLATE FOR A TERMINAL MATRIX

Templates for terminal matrices are page templates with special properties.

X.3.1. GENERAL INFORMATION

Exercise 22-3: Load the current template for the terminal matrix and modify it as desired.

1. Graphical Lists
2. Terminal Matrix
   Right-click
3.CO **Load Page Template**
   The current template of the terminal matrix has been loaded.
4.CA **General □ Select □ All**
5.CA **Edit □ Actions □ Explode**
   Now you can edit the template.

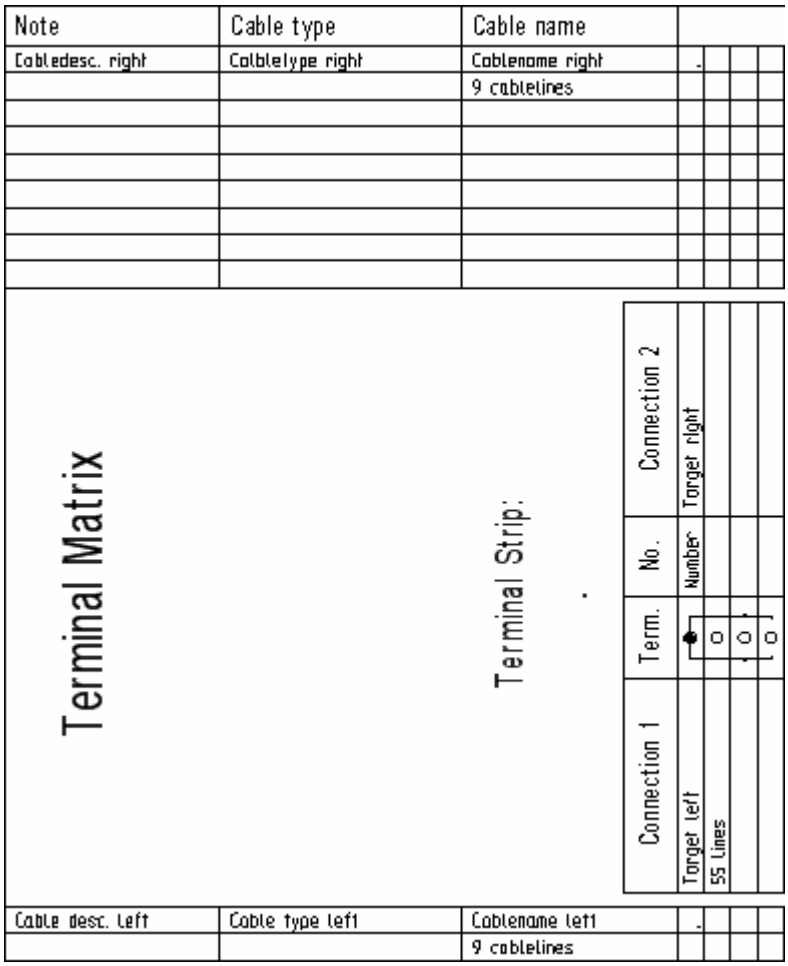
General approach for creating a terminal matrix:

- Create the graphics.
- Define text placeholders by choosing the **Draw □ Elements □ New Text** command.

You need the following text placeholders (available within *Attribute* / *Other node*):

Workspace name
Terminal Sheet
Date
Terminal Sheet Index
Function
Location
Terminal strip

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



Position of the left target of the terminal (for example, Target left behind Connection1)
If you enter here "+P1", a potential appears as a target only at the first terminal, if several terminals are linked. If you enter another text, for example "Target left" as shown in the picture above, then each potential appears at each terminal for linked terminals too, as in the circuit diagram.
If you type "*NoPot*" as text, potentials or reference symbols don't appear as targets for terminals.
If the text placeholder for terminal number doesn't begin with a capital letter, a terminal number appears in each line where there is information about this terminal. If the text placeholder begins with a capital letter, the terminal number appears only in the first line.

Position of the right target of the terminal (for example, Target right behind Connection 2)
Number of lines available for terminals (for example 50 lines), the distance between the position text of the left target and the number of lines defines the distance between the third and next lines for terminals in the terminal matrix.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

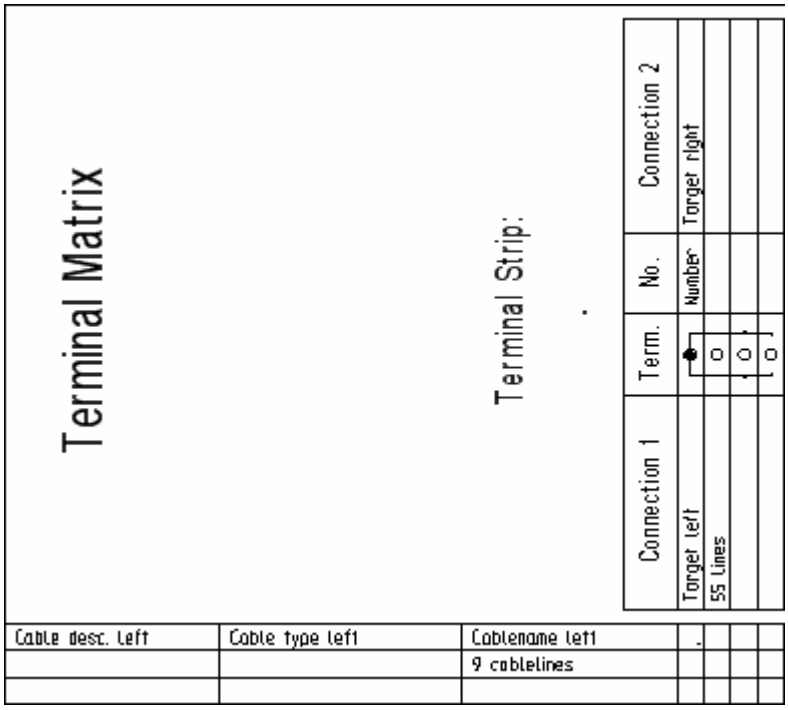
Sheet (and Sheet Index) where the terminal is located
Text placeholder for the function and location of the page
Column where the terminal is located
Terminal type
Terminal description

In case you need reserve terminals, place the text "*Reserve terminals*" in the text placeholder for the left target. The text with an attribute *Spare-terminal* defines the target in the terminal matrix, if a reserve terminal is entered.

There is a feature for defining the index of the first and the last terminal in the page template. If the first available terminal in the Circuit diagram does not possess the index of the first terminal in the page template, reserve terminals are filled up to the first available index. After the last terminal, reserve terminals are filled up to the specified index.
To use this function, a text placeholder with the "normal text" property must be defined as follows:

#Spare +FirstIndex=<n> +LastIndex=<m> +N="<Reserve terminal> <Format Number>"

The text is inserted at the place where the terminal number for terminals in the Circuit diagram is located. No entry is made in the targets of the terminal left or right. (For more details, see chapter "*The Graphical Lists*" in the Help topics).



Cablename left (if you type a dash "-", cable-name, cable-type and cable-description are not displayed)
Cable type left (Type1 – on the left) (if you type a dash "-", the cable-type is not displayed)

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Cable-dimension left (if you enter an article number as a cable-type, you can type the text, such as NYM-J 5x1,5 in the cable-dimension)

Cable desc. left (on the left) (if you type a dash "-", the cable-description is not displayed)

Cable-core Number left, for example c- (on the left)
The texts with the attributes "Cable-core umber Left" (ID=180128) and "Cable-core Number Right" (ID=180131) are used to display a lot of information.
The first letter in the text allows you to define the information to be shown for a cable-core and the second letter allows you to set the text to be shown for a wire:

- displays nothing
+ displays the number (default)
N displays the number (default)
C displays the colour (if existing)
c displays the colour (if existing) and the number (of the colour does not exist)
E always displays the colour (also when number is not used)

If you need more information, you can position a text with the "Cable-core Number left" and "Cable-core Number right" attributes a second and a third time and thus display the information for signal type and signal name.

Q displays information about the signal type
P displays information about the potential name

Cable-core section left (on the left)
The cable-core section can be entered for cable-cores and/or wires, too. Two signs are needed here again. The combination Q- creates the section for cable-cores but not for wires, -Q creates the section for wires but not for cable-cores. For the Numbers of cable-lines left, the same rules apply as for the numbers of lines for terminals.

The following characters determine the appearance of the text:
- text is not shown
* text is always shown
+ cable-core or wire square is shown only if cable-core or wire number is not empty.
Example: -* -> cable core square is not shown, wire square is always shown.

Cable-name (right)
The same rules apply as for the cable texts left

Cable-type right
The same rules apply as for the cable texts left.

Cable-description right
The same rules apply as for the cable texts left.

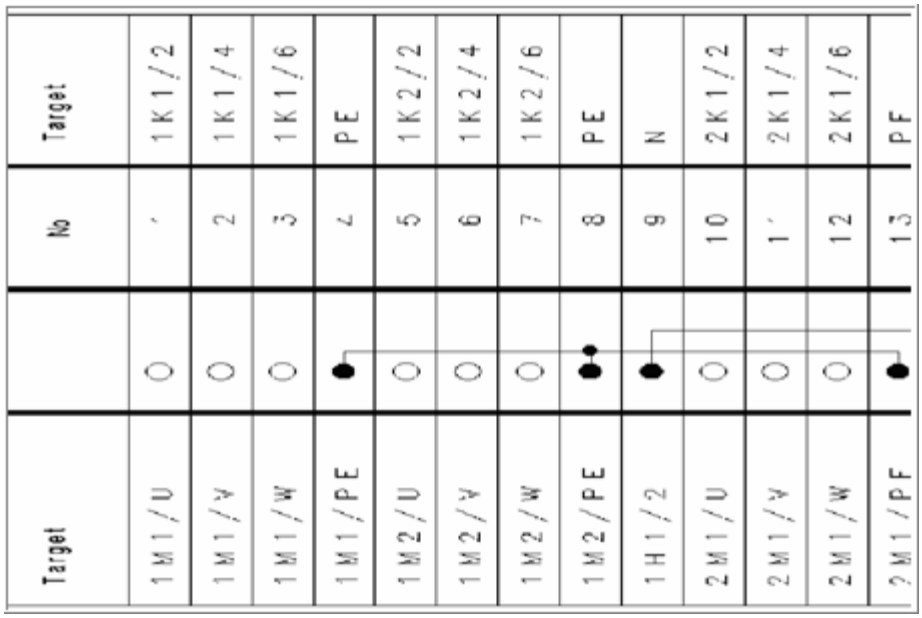
Cable-core Number right
The same rules apply as for the cable texts left.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Cable-core section right
Numbers of cable-lines right, (the same rules apply as for the number of lines for terminals)

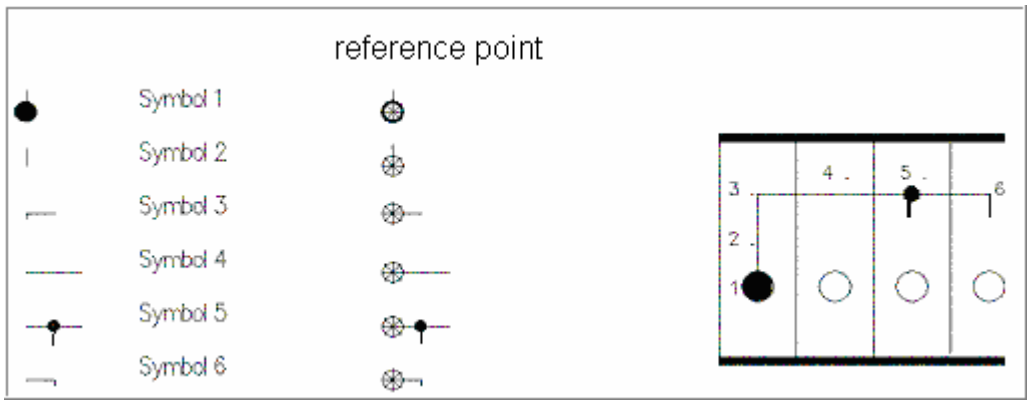
Additional texts can be entered manually.

Define the text placeholders for the bridges.



You need:

- ✓ *one text placeholder (with the "LINK Type" text attribute);*
- ✓ *If the bridge type is not in between 1 to 10: if this text contains the letter P, then bridges identified via potentials are displayed; 0 => bridges identified via wires are displayed;*
- ✓ *1 to 10: bridges of the appropriate LINK Type are shown)*
- ✓ *6 symbols*



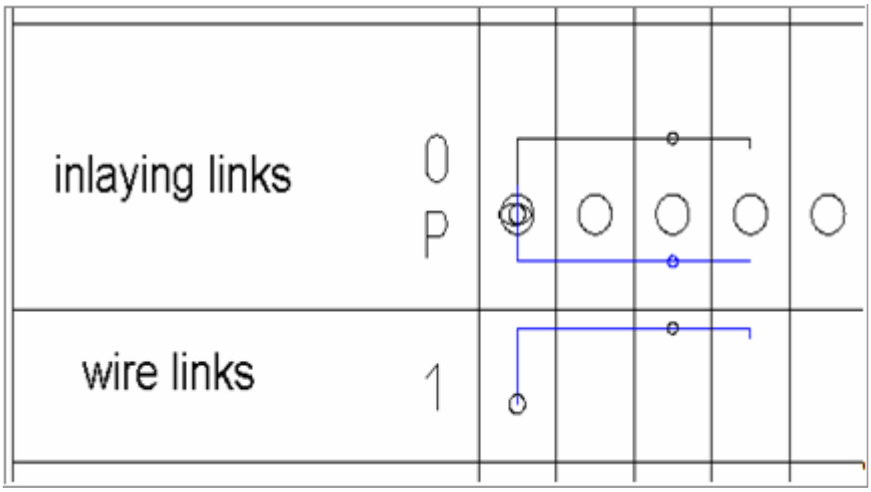
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Create the symbols as follows:

1. Draw the graphics
2. Select the graphics of the symbol
3. Block to a symbol LINK Type 1 or LINK Type 2 or ... LINK Type 6
4. Drag the symbol into the symbol database
5. Delete the graphics used for the symbol
6. Pull the symbol out of the database and insert it.

After you have created and inserted all symbols:

1. Select 6 symbols and the text for the bridge
2. Block as Macro/Group
3. Move the symbol into the symbol database
4. Delete the graphics
5. Pull the symbol out of the database and insert it.



For each LINK Type, you must create a text placeholder (see the picture above - for example, bridge placeholders for P, 0 and 1). If a LINK Type is not defined in the template, then a LINK Type 0 is used.

If a LINK Type P is not placed, bridges identified via wires are not displayed. In this case, the target terminal is shown.
Here, you cannot use all of the settings options available via the template. For more details see chapter "Graphical lists" in the User Manual.

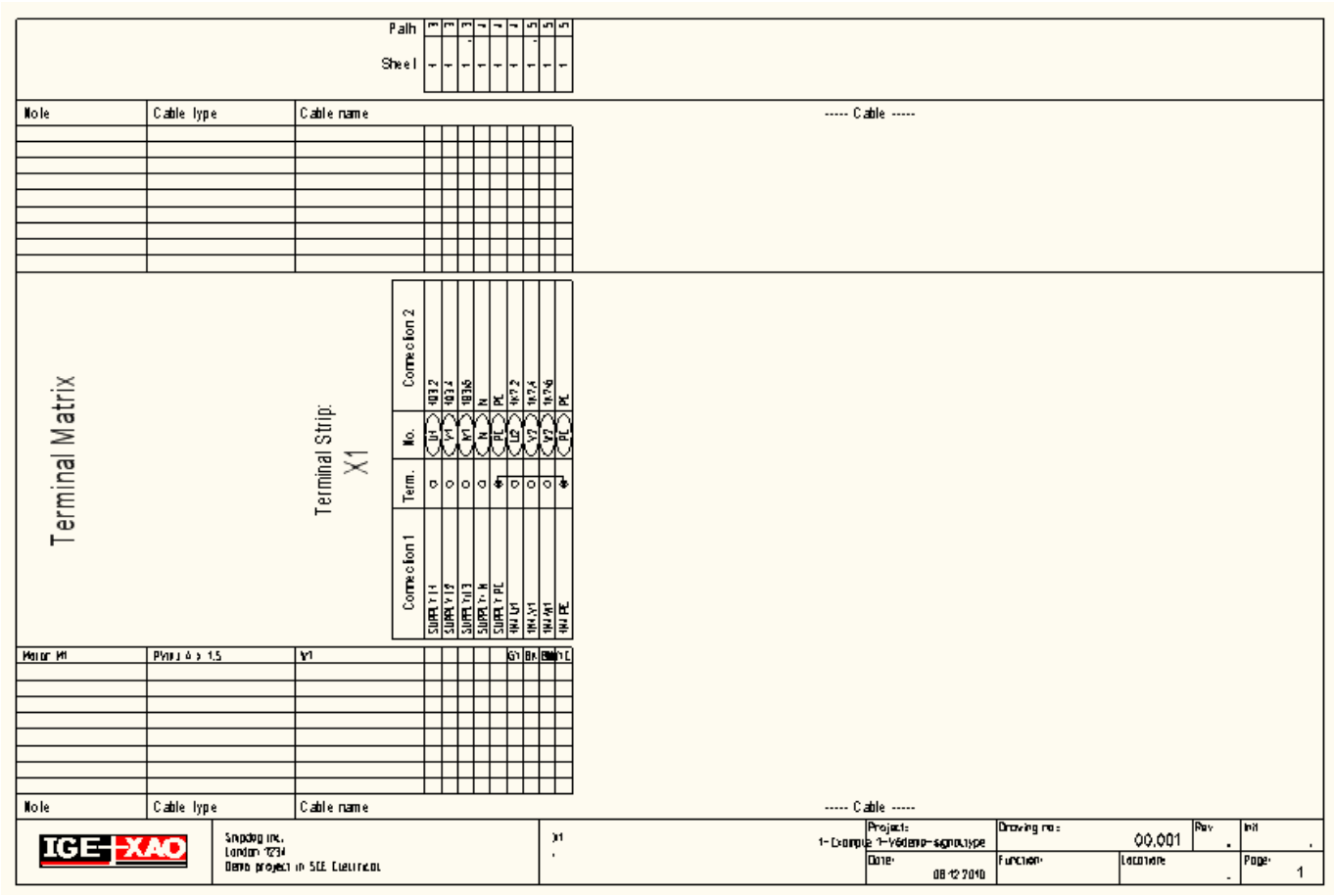
Exercise 22-3: Save the template.

1.CA **General □ Select □ All**
2.CA **Edit □ Actions □ Block**
Group the elements as Page Template, Title block symbol.
3.CO **File □ Save as... □ Page Template**
Define a name for the template and save it.
Set the new terminal matrix as a template in the properties of Terminal matrix.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

X.3.2. DRAWING GRAPHICS ASSOCIATED WITH EACH TERMINAL – ONE GEOMETRY FOR ALL TERMINALS

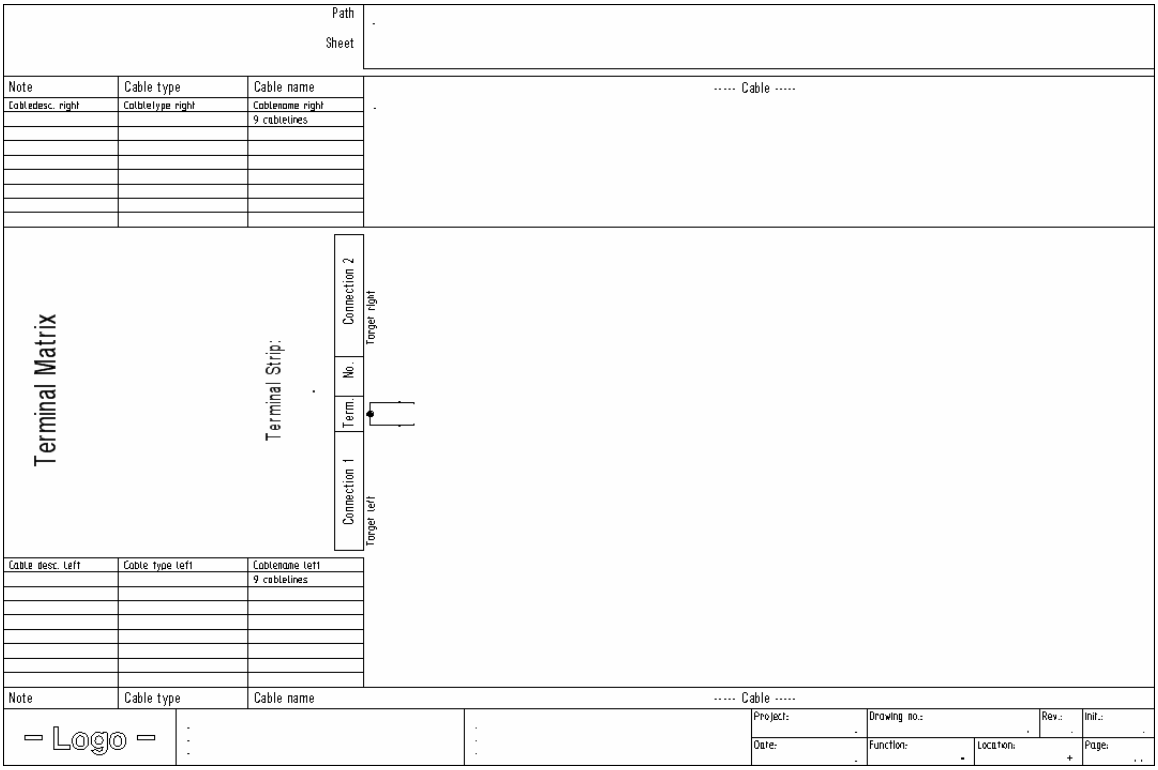
This allows you to draw the geometry for all terminals belonging to the terminal strip.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Generating the page template:

1. Position all the elements of a terminal matrix or a terminal plan, as already described. Do not position the place holder for terminal number. (You can use an existing template and delete the geometry/texts that you do not need.)



2. Generate a *Block/Macro/Group* symbol from all these elements (except for the terminal number, if there is one).
   Generating the geometry for each single terminal:

3a. Draw the geometry for the first part of the terminal.

Group all the elements as a "*Terminal, Line-Block: start*" symbol. If the circle/ellipse for the bridge shall appear only once in a terminal, please add it to the "*Terminal, Line-Block: start*" symbol.

2H1:X1	O	13	X2:14
.		.	X2:13
2H3:X1	O	14	2S3:12
2H4:X1	O	15	X2:16
.		.	X2:15

3b. Draw the geometry for the last part of the terminal.

Group all the elements as a "*Terminal, Line-Block: end*" symbol.

3c. Draw the geometry for the middle part of the terminal (these elements will eventually need to be extended).

Group all the elements as a "*Terminal, Line-Block: trim*" symbol.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

3e. If you want, you can draw geometry for the terminal number. This part will contain the geometry for the bridge, in case the circle appears in each line or if a terminal occupies two or more lines.

Group all the elements as a symbol of the type "**Graphical symbol**".

2H1:X1		13	X2:14
.		.	X2:13
2H3:X1		14	2S3:12
2H4:X1		15	X2:16
.		.	X2:15

3f. Position the text for terminal number.
3g. Group the four symbols mentioned above together with the text for terminal number as a "*Block/Macro/Group*" symbol.
4. Group all the symbols (the macro/group for the standard sheet and the macro/group for single terminals) as a "*Page Template, Title block*" symbol.
5. Save the new page template.

X.3.3. DRAWING GRAPHICS ASSOCIATED WITH EACH TERMINAL – ONE SPECIFIC GEOMETRY FOR EACH KIND OF TERMINAL

Unlike the situation described above, in this case the graphics for the terminal is not taken from the page template, but from a symbol database.
Each terminal can be drawn with the help of a specific graphical symbol (stored in the symbol database), so you can use different symbols for the different types of terminals (for example, spare terminals, end and separator plates).

Terminal Strip:

X2

Connection 1	Term.	No.	Connection 2
S1:3		1	L3
S1:4		2	K2:21
S2:3		3	L3
S2:4		4	K1:21

The symbol for the terminal in the terminal plans has to be assigned to the single terminals, as described below in the part "Which symbols are used for the terminal?"

How to make a new template from an existing terminal plan template:

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Remove all graphics and texts associated with the terminal symbol, except for the terminal number (this text marks the position of the first terminal symbol).

Add a normal text defining a default terminal symbol. The syntax is:

#LineSymbol="<symbol database name>\<folder name>\<symbol name>"

For example:
#LineSymbol="Graphical List\TerminalPlan\0Terminal"

Generating a terminal symbol for the graphical plan (to store in the symbol database)

The symbol needs to contain the graphics of the terminal symbol, as described above (graphical symbol (Id 180180)) and the following placeholder texts:

Terminal number (Id 180112) – this text is obligatory
Terminal connection text left (Id 180152) and right (Id 180153) – optional
Terminal type (Id 180140) – optional
Terminal description (Id 180142) – optional
Terminal free texts 1, 2 and 3 (Ids 180180, 180181 and 180182) – optional

And the page reference texts:

Terminal Page Function (=) (Id 180146), Terminal Page Location (+) (Id 180148), Terminal Sheet (Id 180115), Terminal Sheet Index (Id 180143), Terminal Cell (Id 180116) – optional

Note :

The size of the terminal symbol graphics has to fit the line distance given for the terminal plan (distance between the text for "Target left" and the text for "Number of lines for terminals"). The size and distance must be the same.

Where is the symbol placed in the terminal plan?

In the template, the position of the first line is marked by the "*Terminal number*" placeholder text (Id 180112).

If the terminal symbol has a normal text with the contents "#SybPos", this text is put on top of the placeholder text "Terminal number" (Id 180112) in the template (including the line offset). If no "#SybPos" text is found in the terminal symbol, the "Terminal number" placeholder text from the template is used to position the terminal symbol.

The search is faster if the "*Terminal number*" placeholder text in the symbol contains the "*#SybPos*" text.

Which symbols are used for the terminal?

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

The symbol name for the symbol used in Terminal Matrix, Terminal Plan and Terminal Row Picture Plan can be specified by the "*Terminal Plan Symbol*" component text in the circuit diagram or inside the *Type* database by the "*Terminal Plan Symbol*" type property (Id 12021300), in case you wish to use the same terminal symbol for all terminals of one type. Define the symbol name without specifying a symbol database and folder. The name of the symbol database and folder are given in the page template in the #LineSymbol text.
For spare terminals, the default terminal symbol is used.

Hint

In the advanced level you can define spare terminals via the Terminal Editor.

Inserting headers and endplates

You can insert headers and endplates, if necessary.
The default symbols for header and endplate are defined in the page template. Add to the #LineSymbol= placeholder an additional text with:

+P="<name of symbol for template>"
+H="<name for symbol for header>"

Example of an entry in the Terminal Plan template:
#LineSymbol="Graphical List\TerminalRowPicture\0Terminal" +H="0Header" +P="0Plate"

Plates and spare terminals from the "Component without graphics" list (*advanced*)

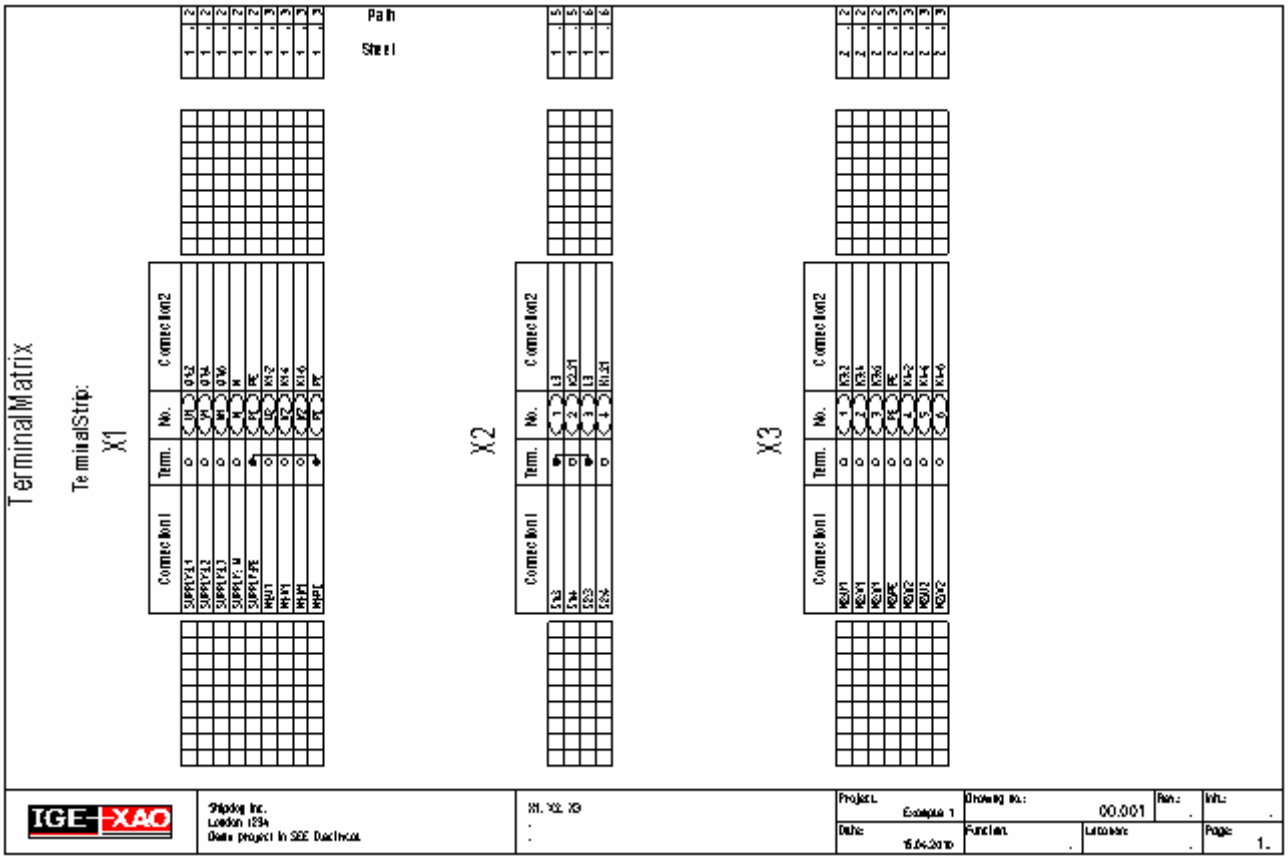
A plate is similar to a terminal and you can manage it as a terminal without graphic (manual component).

The symbol name of a plate can be specified in the same way as for a terminal – by using the "*Terminal Plan Symbol*" (Id 160450) or the type property (Id 12021300).
If the "+P" keyword is available in this text, then the header information ("+H") from the next terminal is used in the Terminal Row Picture plan. Please note that if no header is specified, this means that the default header is used. If no symbol is defined, the default plate symbol is used instead of the default terminal symbol.
This means that "+P=0Plate" is the same as "+T=0Plate +P"

You can also define spare terminals with the Terminal Editor. If you want to add this to the plans, you have to use the "*Spare terminal text*" keyword. Replace the "*Spare*" keyword, in case you have used it to manage spare terminals up to now. If not, please refer to the text above about how to use the "*Spare*" keyword and the "*Spare terminal text*" keyword instead of this. The difference between the "*Spare*" and "*Spare terminal text*" keywords is that exactly the terminals defined in the ***Terminal Editor*** are used as spare terminals.

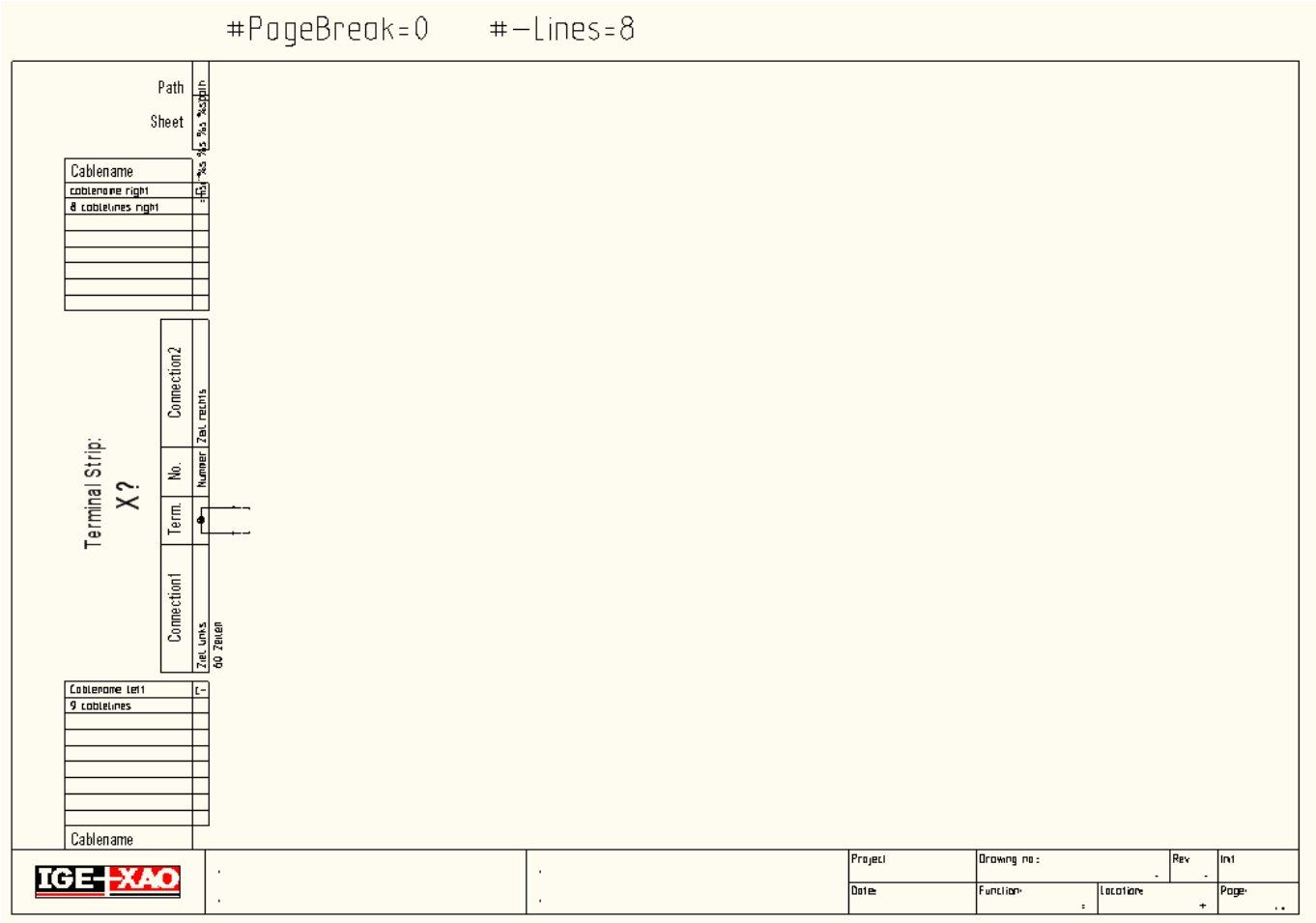
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

X.4. DRAW MORE THAN ONE TERMINAL STRIP PER PAGE



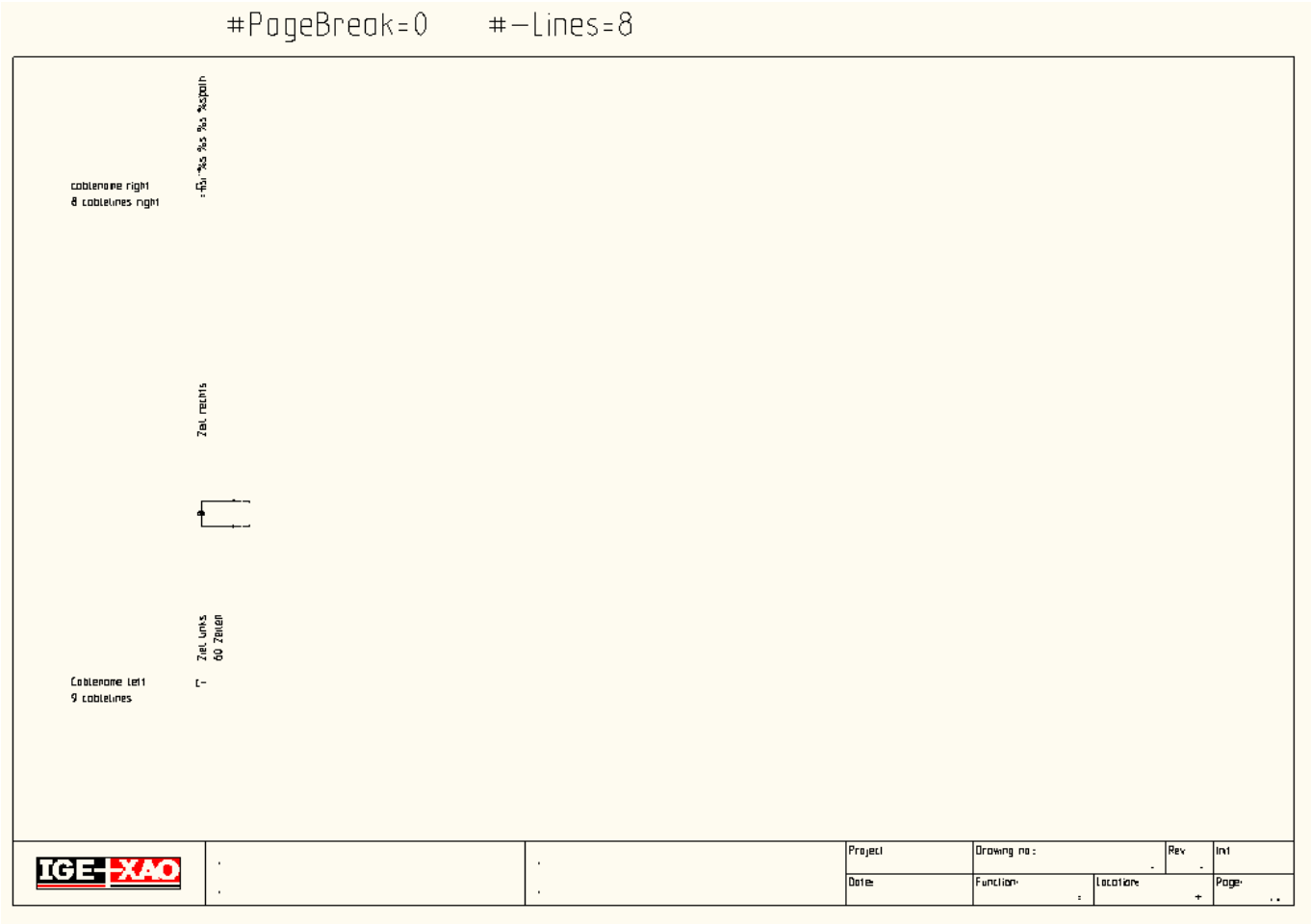
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Generating the Page template:



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

1. Insert the elements of a terminal matrix or a terminal plan as described. Do not insert the placeholders for the terminal strip and number. (You can use an existing template and delete the geometry/texts that you do not need).



2. Add a "Normal" text attribute to the content.

#PageBreak=0
or
#PageBreak=0 #Lines=8 (for example)
where #Lines=8 indicates that the terminal strip header will use the space for eight terminal lines (if you do not use any #Lines string to specify the terminal row header, the space will be calculated from the graphics in the template.)

The text "#Pagebreak" in the template can be used to control the generation of the new page.
#PageBreak=0
Multiple terminal strips are generated on one page like before, a new page is generated only when the previous page is already filled.

#PageBreak=1 or #PageBreak=160010 or #PageBreak="-"
are used to start a new page f the name of the terminal has been changed.
#PageBreak=140020 or #PageBreak="="
are used to start a new page if the function (=) has been changed.
#PageBreak=140050 or #PageBreak="+"

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

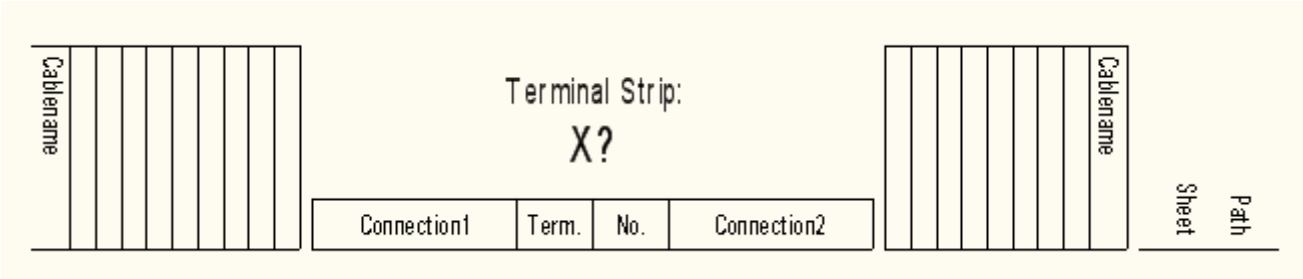
Are used to start a new page if the function (+) has been changed.

Additionally, you can define how to manage the function and location information in the terminal strip. If you add the text +DL0 to the #PageBreak command, the terminal strip name is always extended with the function and location information (even if the terminal strip has the same function and location information as the page where it is located).

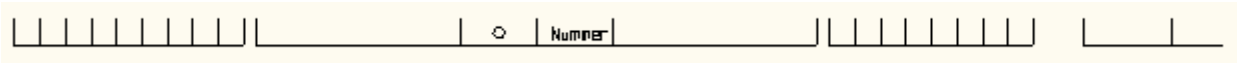
If you add the text +DL1 to the #PageBreak command, the terminal row name will contain only the function and location information in case they are different from the ones on the page.

3. Generate a *Page template* symbol from the inserted elements, except for the terminal number and terminal strip name.

1. The terminal row header part has to be created in the following way
All the geometry and texts that show information that is never changed have to be groupd as a "Graphical symbol". The graphical symbol and a text with the attribute "Terminal number" have to be grouped as "Macro/"Group".



5. All the geometry and text that show information about the terminal that is never changed have to be grouped as a "Graphical symbol". The graphical symbol and a text with the attribute "Terminal number" have to be grouped as "Macro/"Group".



6. Group all elements as a symbol of the type "Graphical symbol"

2H1:X1		13	X2:14
.		.	X2:13
2H3:X1		14	2S3:12
2H4:X1		15	X2:16
.		.	X2:15

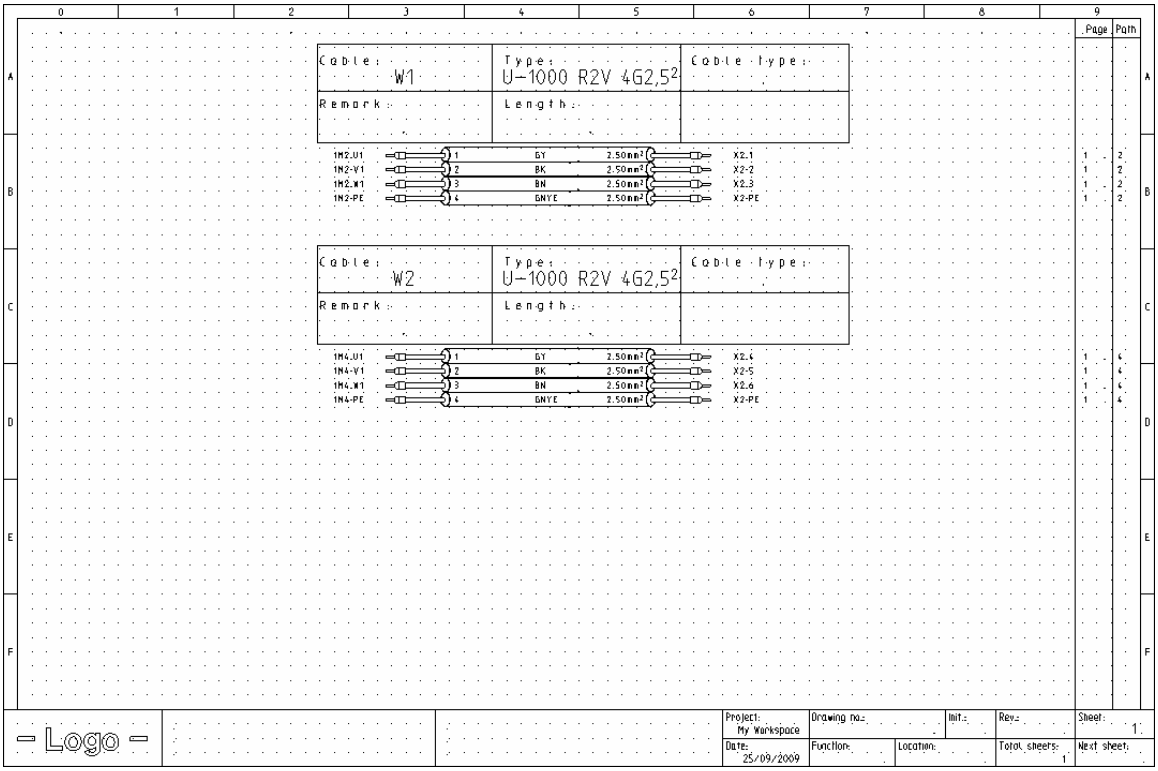
7. Insert the text for the terminal number
8. Group the four symbols and the text for the terminal number as a "*Block/Macro/Group*" symbol.
9. Group all symbols (the macro/group for the standard sheet and the macro/group for the single terminals) as a "Page Template, Title Block" symbol.
10. Save the new page template.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Y CABLE PLAN WITH GRAPHICS

(standard)

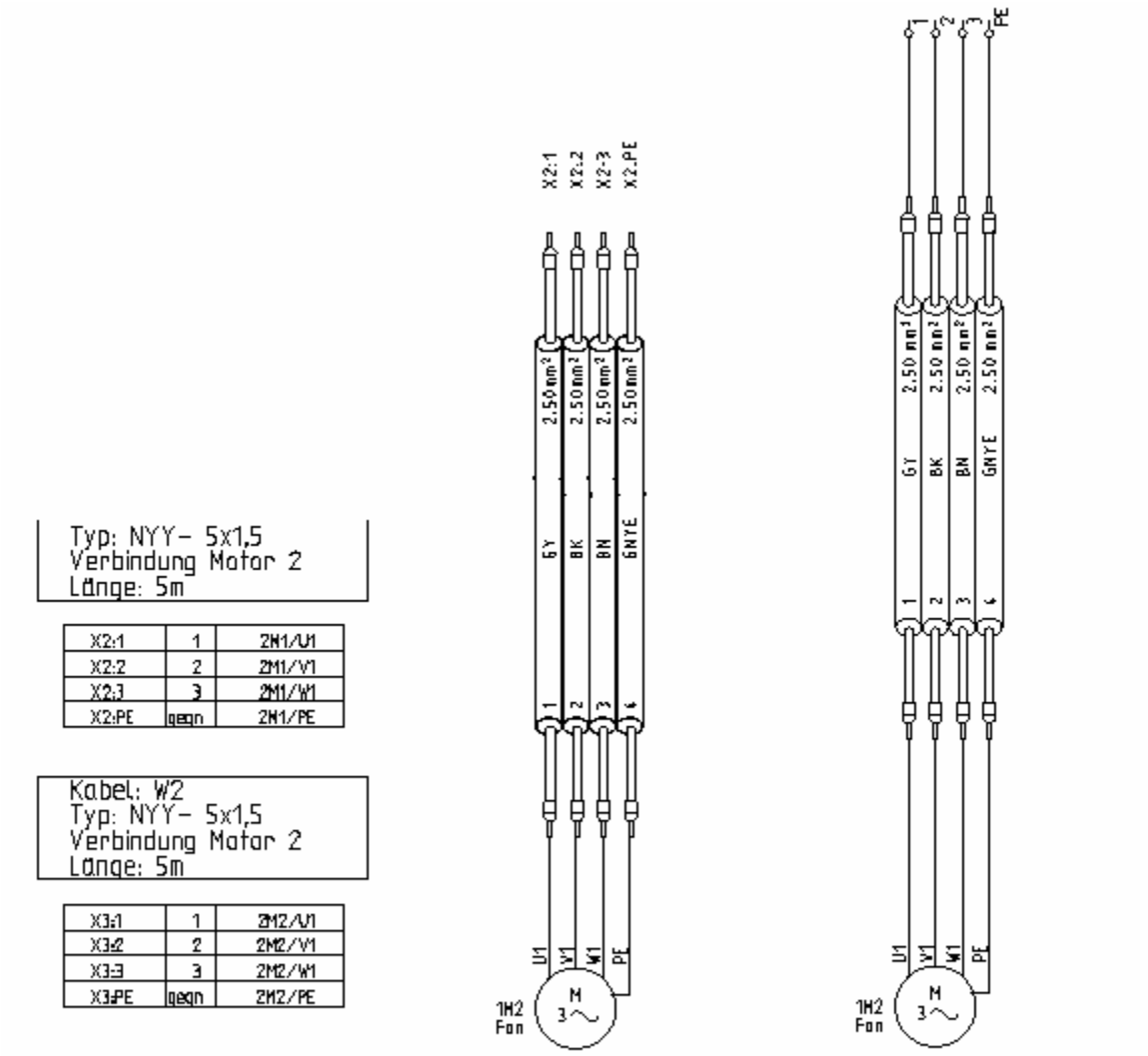
The *Cable plan* with graphics enables a clearly-presented documentation of the cable cores and their targets.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

The targets of the cable-cores can be displayed as text information or graphics.

Examples:



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Y.1. CREATING A CABLE PLAN WITH GRAPHICS

Exercise 23-1: Create the *Cable plan with graphics* for the training workspace.

1. Select **Cable plan** within the ***Graphical lists*** area in the Workspace tree.

2. Right-click with the mouse.
3.CO **Generate**
The *Cable plan with graphics* is generated.

Exercise 23-2: Viewing the *Cable plan*.

1. 0001
Select page 1 of the Cable plan with graphics by double-clicking on 0001 beneath *Cable plan* in the *Workspace tree*.

Y.2. CREATING A TEMPLATE FOR CABLE PLANS WITH GRAPHICS

The template consists of the following parts:

- ✓ *Elements of the standard sheet -> Page Template, Title block symbol*
- ✓ *Texts for first page number and page break (optional) (PageBreak=, + or -)*
- ✓ *Header for Cable (Name, Type, etc.)*
- ✓ *Data about Cable core number*
- ✓ *Target left/ Target right*
- ✓ *Number of lines for Cable cores*

Exercise 23-3: Create a template for Cable plan with graphics.

1. Draw the desired graphics and place the needed texts. Select All. Group the selected elements as a "Page Template, Title block" symbol.
2. Insert texts for the first page number and page break (optional) (PageBreak=, + or -)
3. Create the Header for the cable:
3a. Group the graphics and the text with the "normal text" attribute as a "Graphical Symbol".
3b. Insert the texts for the cables. These are texts with the attribute "Cable name", "Description", "Type", etc.
(Optional: A text of the form "Type: %s" provides the result, for instance Type: NYY 5x1,5.)
3c. Group the "Graphical Symbol" and the texts for the cable as a "Block/Macro/Group" symbol.
4. Define the view of the data for the Cable core number.
4a. Group the graphics and the text with the "normal text" attribute as a "Graphical Symbol".
4b. Insert texts for:
Cable core number (+c (colour/number) or +C (only colour) or + (only number) or – (nothing);
Other cable attributes: colour, cross section, potential name, signal type of wire:

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Position a text with the "cable core colour" or "cable core square" attributes. Type the code given below into this placeholder. The text is replaced by the associated text:

"+S" – cable core square
"+Q" – signal type of wire
"+P" – potential name for wire

Maximum five instances of the text for cable core colour and cable core square can be used, for example three times colour and two times square.
Function, Location, Sheet, Index, and Column where the Cable core is located (optional).

4c. Group the selected elements as a Block/Macro/Group symbol.

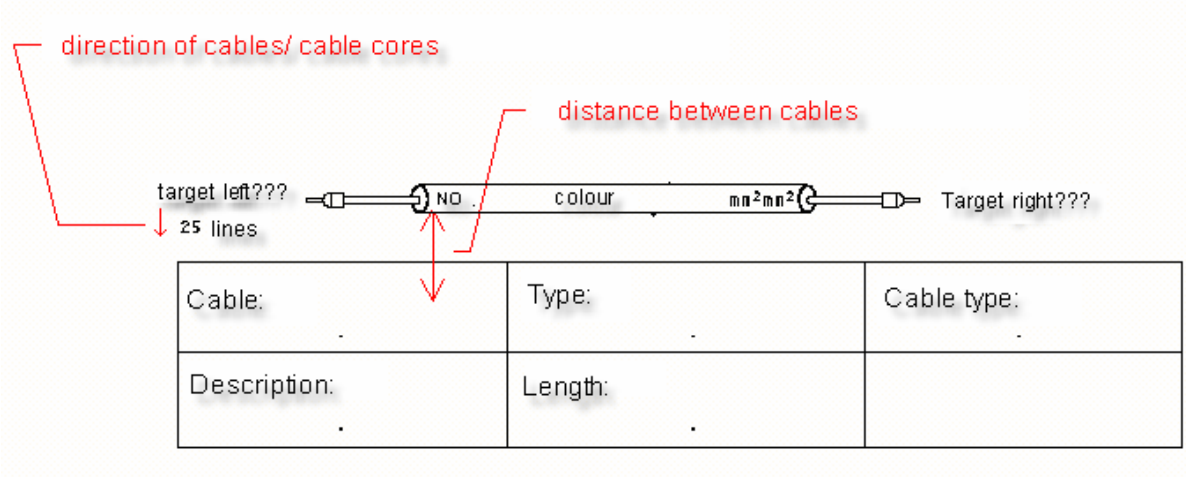
5. Define the targets left and right.

5a. You need texts with the attribute "Target left" (or "Target right").
If you type in +NoSymbol into the text for the target left, no target symbols will be displayed, only the component text is used as target.

5b. If Circuit diagram symbols must be placed, 2 routes and 1 text are required for the position of the symbol (look in the "List of Construction set" symbol library, "Cable plan with graphics" folder).

6. Define the distance between different cables on one page.
You can define the distance via the texts "*Cable core*" in the Cable core information, and "*Cable name*" in the Cable information.

You can define the direction, in which cable and cable cores appear in the template, by means of the distance between the texts "Target left" and "Number of lines" in the Cable core information.



7. Save the page template.

8. Assign the new page template in the Properties of the Cable plan.
Here, you cannot use all of the settings options available via the template. For more details, see chapter "Graphical lists" in the Help topics.

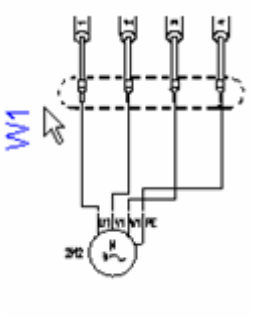
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Y.3. SHOWING NOT USED CABLE CORES IN CABLE PLAN

Spare cores can only be shown, if the cable got a type and the number of cores is defined inside this type.
If you want to show the spare cores, the template must be changed. Just add a text with attribute "normal" and content #Spare or #Spare=1.
If a text with content #Spare=0 is found, no spare cores are added to the cable plan.

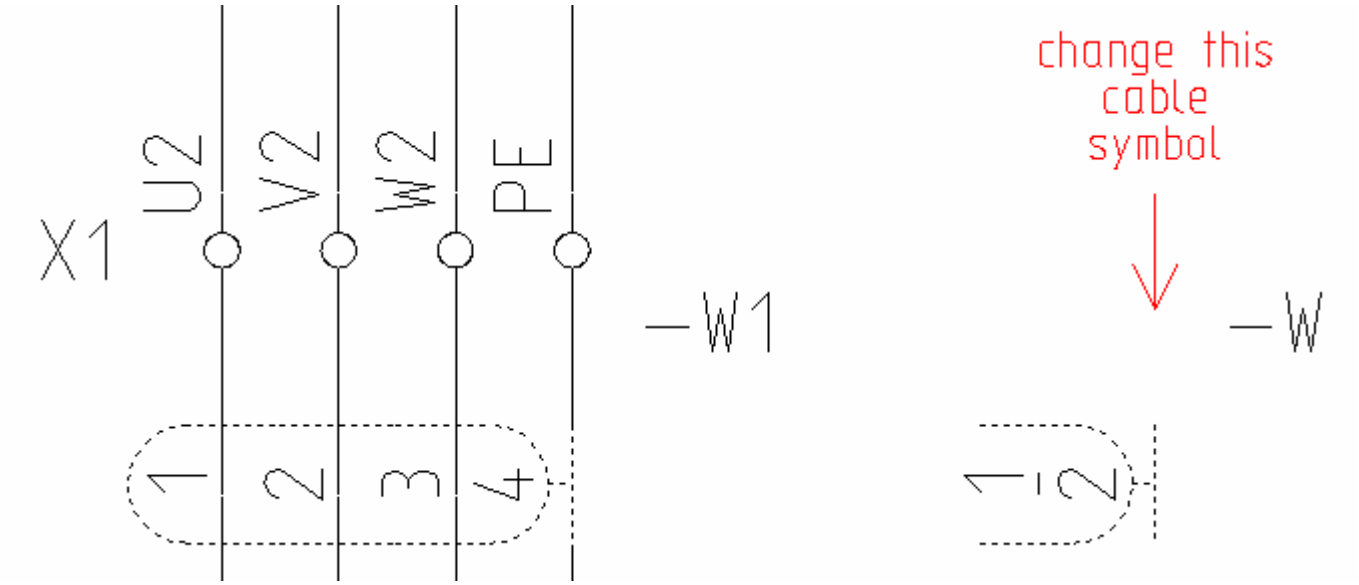
Y.4. CABLE PLAN WITH SHIELD INFORMATION FOR CABLES

If you use user-created cable symbols for shields inside the circuit diagram, you may want to show these symbols in the graphical cable plan as well. This is possible with the following 3 steps:

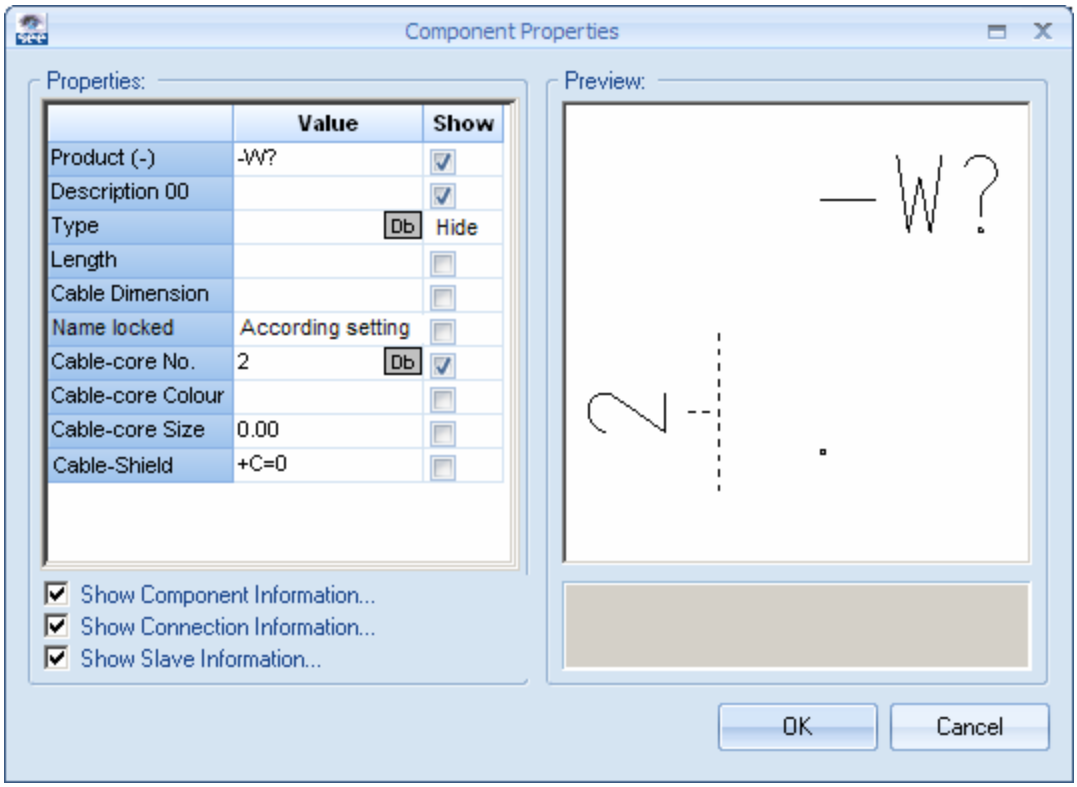


1. Circuit diagram:

Each cable for the circuit diagram needs to contain one symbol with a component text with the "*Cable-Shield*" attribute. (This means, you have to add this text to your cable symbols and then redraw existing cables, if you want to use cables with shield in your cable plans.)
It is recommended that you change the cable symbol which contains the shield:



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



The component text "*Cable-Shield*" can contain the following keywords:
+S="Name of the symbol folder in the "*Cables.SES*" symbol library"

This name replaces the symbol group name in the template placeholder-text "*Cable-Shield*". It is used by all shields of this cable.
-S= Do not draw the shield in the cable plan. (The effect is the same as if this component text does not exist)
+C=<n>
where <n> gives the number of the core which is connected to the shield.
n=1: the first core is connected to the shield
n=2: the second core is connected to the shield
...
n=0: the last core is connected to the shield

2. Template for the graphical Cable plan

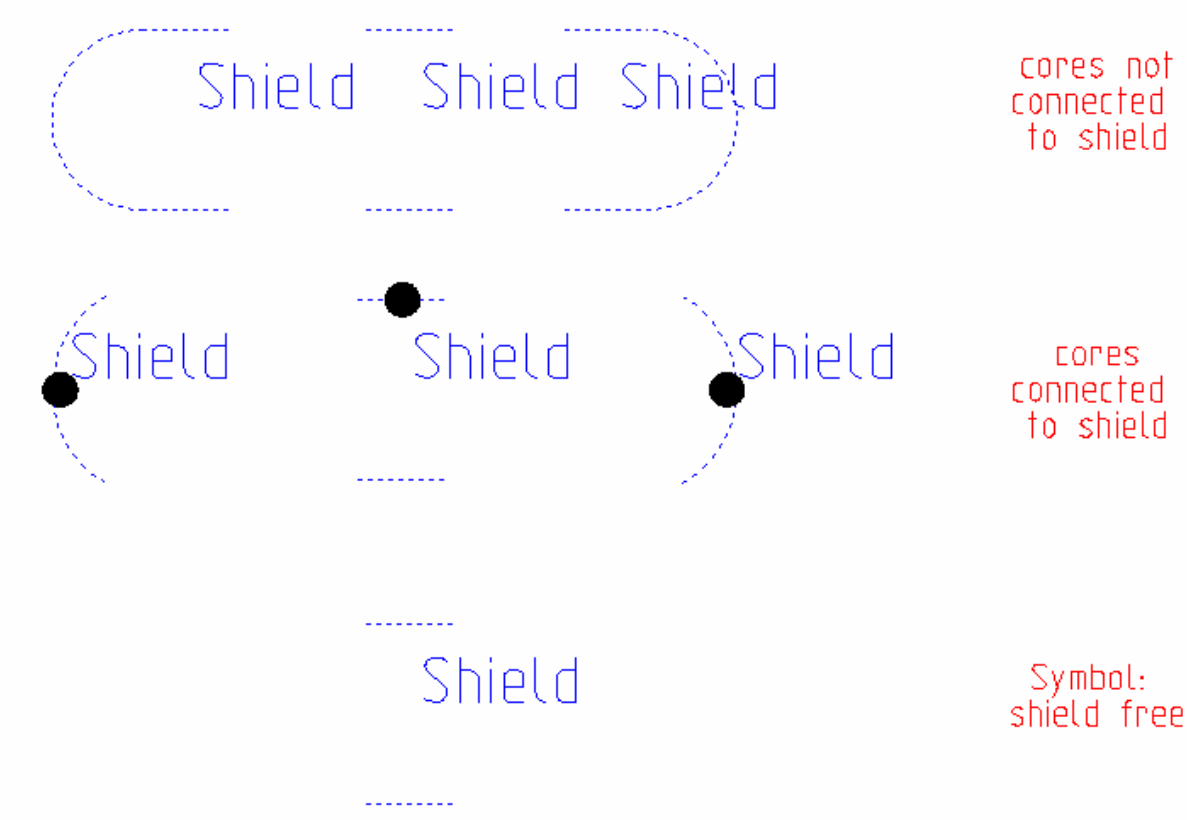
Add one or two placeholder texts "*Cable-Shield*" to the template of the graphical Cable plan.
These placeholders mark the position of the Cable-Shield (right and/or left).
The placeholder-text can contain the following keywords:
+S="Name of the symbol library\folder\symbol name"
Default: +S="Cables\CablePlan\Shield"
If this placeholder exists, the shield symbols defined in the circuit diagram cable symbols are not used.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

2. Generate cable-shield-symbols for the graphical Cable plan

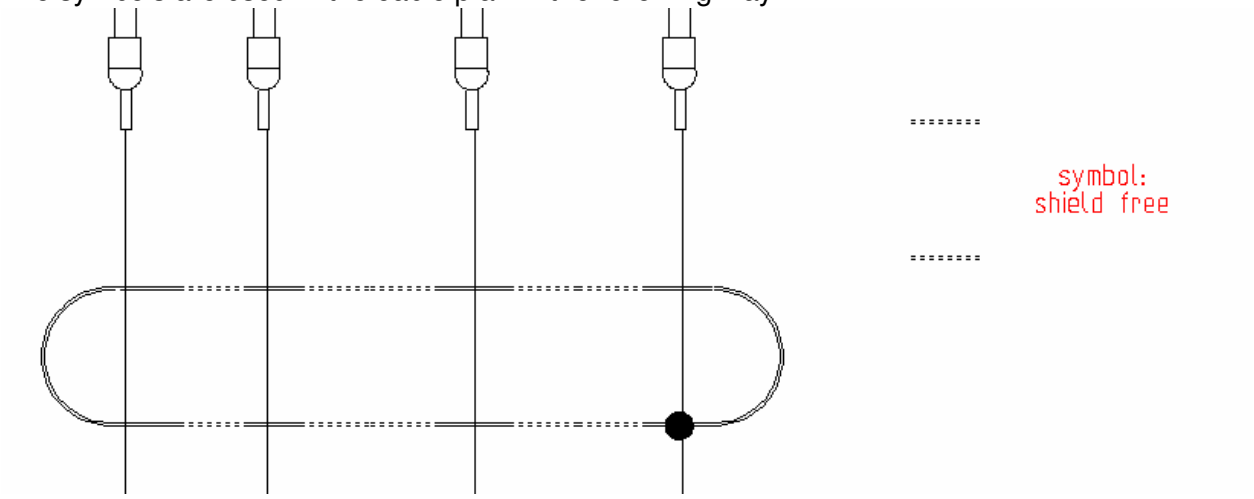
A cable shield in the graphical *Cable plan* is constructed by symbols from a symbol group. Such a symbol group consists of 7 symbols. The default symbol group should be stored in the *CABLES.SES* library in the folder *CablePlan* like follows:

Symbol from the library	Description
Cables\CablePlan\Shield-Start	This is the start symbol if the first core is not connected to the shield.
Cables\CablePlan\Shield-StartC	This is the start symbol if the shield is connected to the first core.
Cables\CablePlan\Shield-Mid	This is the middle symbol if a core is not connected to the shield.
Cables\CablePlan\Shield-MidC	This is the middle symbol if the shield is connected to this core.
Cables\CablePlan\Shield-Free	If there is spacing between two cores in the cable plan, this symbol is used to make the shield look closed.
Cables\CablePlan\Shield-End	This is the end symbol if the last core is not connected to the shield.
Cables\CablePlan\Shield-EndC	This is the end symbol if the shield is connected to the last core.

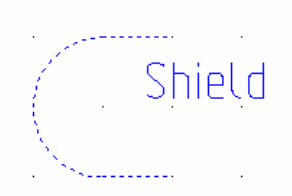


COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

The symbols are used in the cable plan in the following way:



Each symbol consists of geometry and a placeholder-text "*Cable-Shield*", see the text "*shield*" in the picture below:



This placeholder text is necessary in order to place the symbol in the template on top of the text "*Cable-shield*". The text of the second symbol is positioned with a distance to the first one, that is given by the distance between the template-texts "*Target Left*" and "*Number of cable core lines*" (Id= 180114). Each cable shield symbol has to fit into the space given by this distance.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Z WIRING LISTS

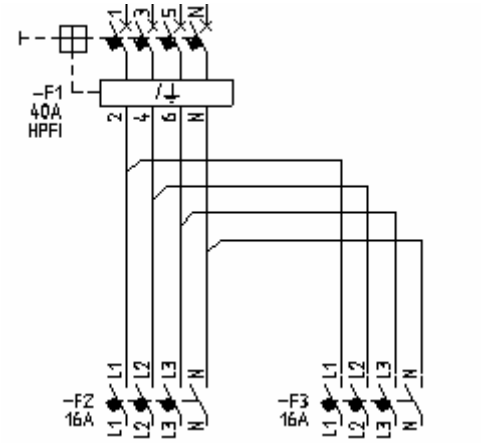
(standard)

The **WiringList** command allows you to generate a wiring list in *Excel* or *Text* (*ASCII*) format.

Term definitions

In the wiring list, the targets for each wire are inserted. The wiring list groups all targets which are on the same potential.
Each wire has properties: wire number, wire colour, wire section, wire type, signal and potential.
If several wires are on the same potential, the wires are grouped with the same wire properties. In this case you can speak of a network.
If the "*Potentials first*" option is active, then in the listing of targets the potential is always first, if it exists.
The wire direction is taken into account when the wiring list is created, but no distinction is made between the start point and the end point of the wiring.

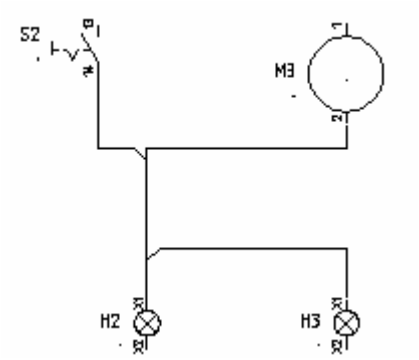
Example:



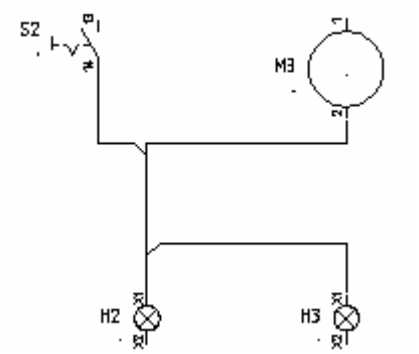
In the wiring list the elements of the first network can appear either in the order "-F1/2; -F2/L1; -F3/L1" or in the order "-F3/L1; -F2/L1; -F1/2".
In some cases two or more networks may appear, even though there is only one network.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Example:



The network "S2:14, H2:X1, H3:X1" appears - it cannot be continued from here. That is why the network "H2:X1, M3:2" appears in addition.



If you modify the wire direction, as shown above, you get: "S2:14, M3:2, H2:X1, H3:X1".

If you use wiring lists, you should not insert any symbols on the potentials, because the potential will appear twice as targets for the equipment.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Z.1. GRAPHICAL WIRING LIST

The graphical *Wiring list* is available in the *Graphical Lists* area of the ***Workspace Explorer***.
The template contains only placeholders with the "*Normal text*" attribute.
For a wiring list specific criteria for sorting are valid, as described below. That is why the #OrderBy, #Where, #PageBreak and #LineBreak commands cannot be used here.

Besides the ordinary placeholder texts in the page template, the following special placeholder texts can be used:

ID	Description
#Lines 30 7.5	It specifies the maximum number of lines (30) and the distance (7.5) between them. This text can be located anywhere in the page. **NOTE:** You can change the number of lines and the line distance.
#180110 $#	Consecutive number for line It is possible to add a prefix (X) or a postfix (z) to each number ($#). If you want to do so, use the following syntax: #180110 x$#z
#175150	Cross-section
#175151	Colour
#175152	Number (of the wire)
#175157	Signal type
#175158	Potential number
#180111	Combination of wire attributes If you want a combination of the wire attributes (number, size, colour, type, signal type or potential name) in one text, please proceed as follows: Combine the text placeholder #180111 in each combination with the placeholders: $N = wire number, $C = wire colour, $S = wire size, $T = wire type, $P = signal type, $Q = potential name Example: #180111 $N $C $S If you want, you can use a separator (for example /). Example: #180111 $N / $C / $S The attributes appear in the Wiring list in the sort order in which you define them in the template. You can add a blank between the different attributes, like shown in the example or a separator, for example a comma.
#180112	Represents the list of targets of each wire part Example: #1801112 #Sep= " / " #Len=170 #Cnt=0 #Sort="1NCS" If you use this placeholder, all targets are listed one after the other. Different parameters allow you to define the appearance. (All parameters must be in the definition.)
#Cnt=	Specifies the maximum number of targets in each line of the text (0 means ignore) You can either use #Cnt or #Len to control the length of the string.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

#Len=	Specifies the maximum length of each line of the text (0 means ignore) You can either use #Cnt or #Len to control the length of the string.
#Sep=	Specifies a separator between the targets (for example /). The separator has to be inserted between inverted commas (" "). With this, you can also add blanks. Example: #Sep=" / "
#Sort=	Controls wire part sorting. Example: #Sort="0NCS" or #Sort="1NCS" The number in that parameter specifies if potentials are set at the beginning of the wiring list or not and if the wiring direction is taken into account. The first character after **"#Sort="** is "0", "1", "2", or "3" and it sets the sorting mode for targets: "0" disables "sort potentials first" + disables "Sort targets by name" (ignore wire directions) "1" enables "sort potentials first" + disables "Sort targets by name" (ignore wire directions) (DEFAULT) "2" disables "sort potentials first" + enables "Sort targets by name" (ignore wire directions) "3" enables "sort potentials first" + enables "Sort targets by name" (ignore wire directions)

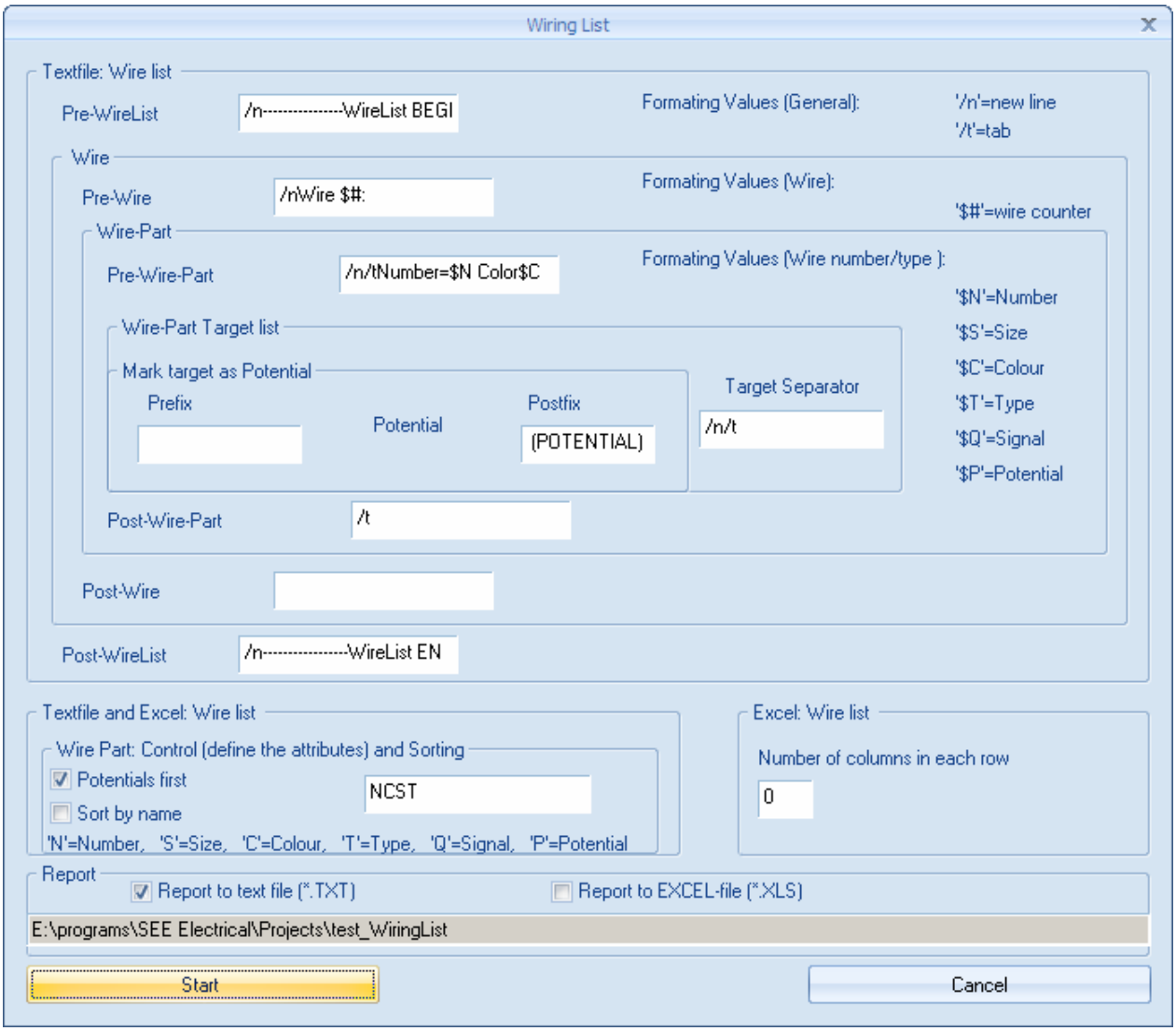
Z.1.1. CREATING AN EXCEL/TEXT FILE

The files are created via the **WiringList** command in the ***Commands*** tab. The command is available only in case a circuit diagram sheet is currently open.

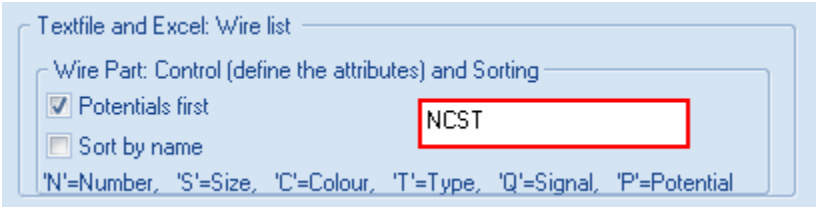
- Execute the command from the ***Commands*** explorer.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

The following window appears:



- Specify, in the area ***Text file and Excel: Wire list***, which information is to be displayed and in what order it is to appear.


Textfile and Excel: Wire list
Wire Part: Control (define the attributes) and Sorting
Potentials first
Sort by name
NCST
'N'=Number, 'S'=Size, 'C'=Colour, 'T'=Type, 'Q'=Signal, 'P'=Potential

In the example above, the information about wire number, wire colour, wire section and wire type will be displayed in the specified order.

✓ *If you tick the "**Potentials first**" option, you can define if potentials should appear at the beginning of the wiring list or not.*
✓ *If you tick the "**Sort by name**" option, you can define whether the wires should be sorted by wire direction or by wire name.*

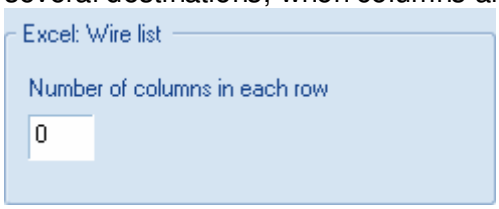
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Excel list

The information in an *Excel* file basically corresponds to the information in the graphical list. Column "***A***" shows a counter number, and the next columns display the wire properties, as defined in the ***Text file and Excel: Wire list*** area. The next column shows whether a signal is present. Then follow the columns with the targets.

	A	B	C	D	E	F	G
1	Counter	Wire-number	Wire-color	Wire-size	Wire-type	Potential	Target
2	001	L1	BK	1,5		L1	F1:1
3		L1	BK	1,5		L1	F2:1
4		L1	BK	1,5		L1	F3:1
5		L1	BK	1,5		L1	-Q1:1
6	002	L2	BK	1,5		L2	F1:3
7		L2	BK	1,5		L2	F2:3
8		L2	BK	1,5		L2	F3:3
9		L2	BK	1,5		L2	-Q1:3

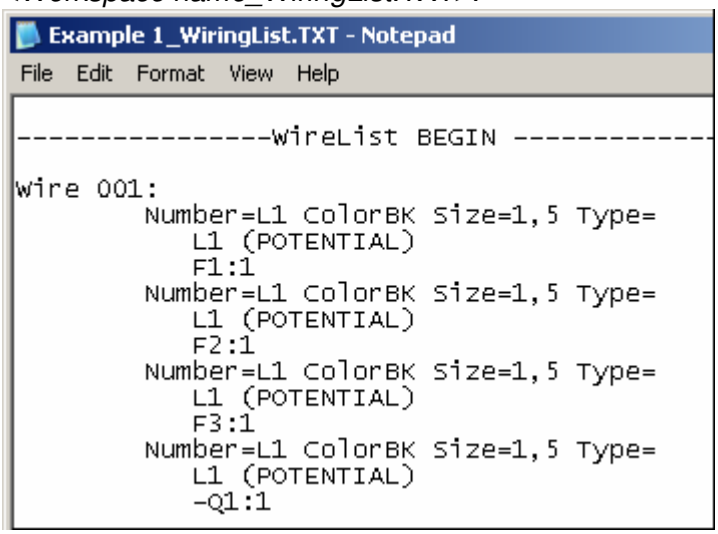
By using the "***Number of columns in the line***" field, you can specify how many columns will be created for targets (=column "Potential", if it is present + columns for targets). If there are several destinations, when columns are available, a new line begins.



By default the *Excel* file is saved in the *\Projects* installation folder. It receives the name *<Workspace name.XLS>*.

Text file (ASCII file)

By default the Excel file is saved in the *\Projects* installation folder. It receives the name *<Workspace name_WiringList.TXT>*.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

The following formatting options are available:

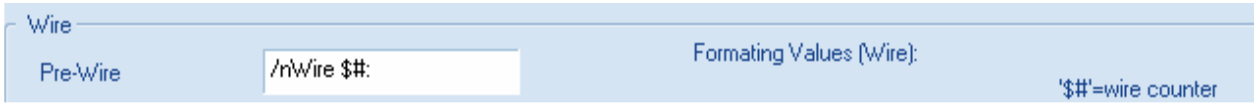
Pre-Wire list/Post-Wire list

In this field, you have the possibility to define a title for the wiring list.

- ✓ */n defines the beginning of a new line*
- ✓ */t defines if the values of this area should be inserted with a tabulation to the right*

For the *Post-Wire* list the same rules apply. If a line is not desired, leave it empty.

Pre-Wire/Post-Wire

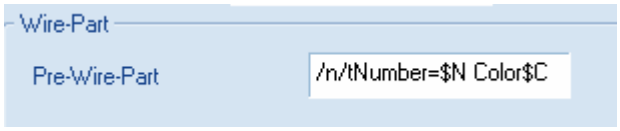


In this field, you have the possibility to specify a title for the wire network.

- ✓ */n defines the beginning of a new line*
- ✓ */t defines if the values of this area should be inserted with a tabulation to the right*
- ✓ *$# displays the order number.*

For the *Post-Wire* the same rules apply. If a line is not desired, leave it empty.

Pre-Wire Part/Post-WirePart



In this field, you have the possibility to specify a title for a network part (= wires with the same wire properties).

- ✓ */n defines the beginning of a new line*
- ✓ */t defines if the values of this area should be inserted with a tabulation to the right*
- ✓ *Various placeholders allow you to set the desired values: $N (=wire number), $C (= wire colour), $S (=wire section), $T (=type), $Q (=signal) and $P (=potential).*

For the Post-Wire-Part the same rules apply. If a line is not desired, leave it empty.

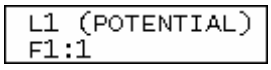
Mark Target as Potential

In this field, you have the possibility to insert a prefix or a suffix which indicates potentials:

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

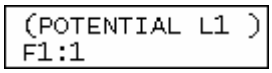
Example 1:

Result:



Example 2:

Result:



Target Separator

In this field, you have the possibility to define whether the targets will be displayed in a single line or not.

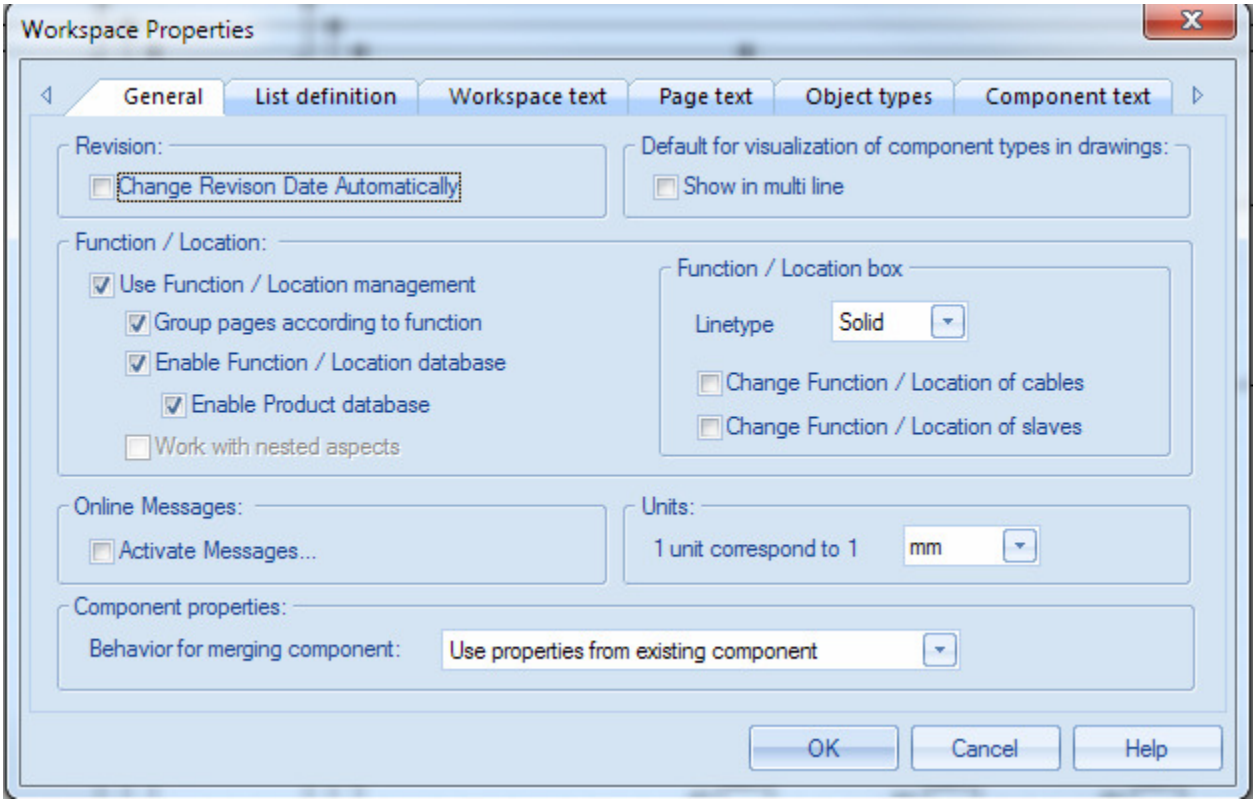
- ✓ */n defines the beginning of a new line*
- ✓ */t defines if the values of this area should be inserted with a tabulation to the right*
- ✓ *If you type in a character as separator, for example a semi-colon, all the targets will appear on a single line.*

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

AA FUNCTION AND LOCATION

(Standard)

In *SEE Electrical Standard* or *Advanced*, you can choose to work with function/location by activating the settings in the ***Workspace Properties*** window.



If you work with function and location, a component name consists of three parts: function (begins with =), location (begins with +) and component name (begins with -). In most cases the function and location are transferred from the legend of the standard sheet.

Exercise 25-1: Create a new workspace.

1.CA **File**
2.CO **New**
3.> File name
Type in a name for the new workspace.
4.> **Save**
5.> Choose the workspace template.
6.> Click **OK** to confirm.

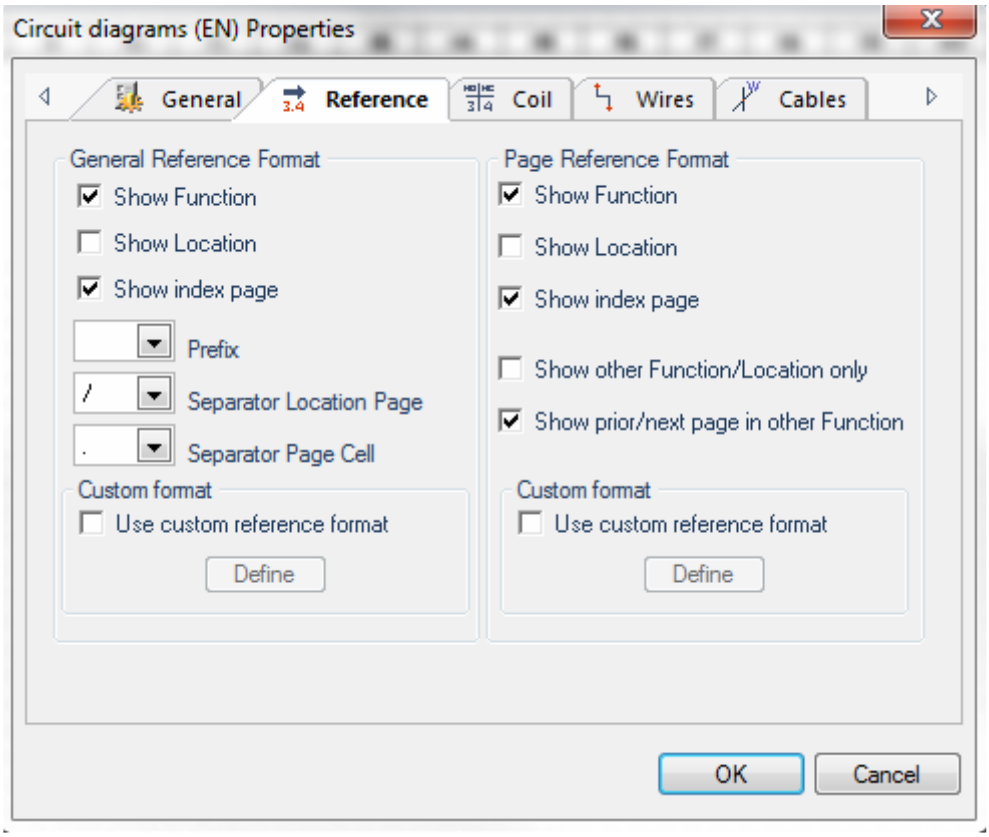
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercise 25-2: Change the properties of the workspace. You are going to work with function and location in your new workspace.

1. Click the workspace name in the ***Workspace Explorer***.
2. Right-click with the mouse
3. CO **Properties**
   The ***Workspace Properties*** window appears.
4. > Use **Function/Location** management
   Tick the option. The "***Group pages according to function***" option is automatically activated. Leave it ticked.
5. > Choose the linetype to use for the function/location box.
6. > **OK**

If you have chosen to "***Group pages according to function***", you must change the settings for the generation of cross refereneces as in this case you might have duplicated page numbers so the function must be included in cross reference too.

7. Click *Circuit diagrams* in the *Workspace Explorer*.
8. Right-click with the mouse.
9. Open the ***Refrences*** tab.



10. Enable the "***Show Function***" option.
11. **OK**

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercise 25-3: Create page 1 of the workspace and enter a function/location in the Page information dialogue box. The values appear also in the standard sheet of page 1.

1. Click *Circuit diagrams* in the Workspace Explorer.
2. Right-click with the mouse
3.CO **New**
   Type the function and location data in the **Page information** dialogue box.
4.> Function
5.# A1
6.> Location
7.# 01
8.> **OK**

Create pages 2 and 3 in the same way. Enter function/location for page 2, too.
Give a different identification for page 3.
You can see in the workspace tree that the workspace has been restructured according to the function and location.

Exercise 25-4: Copy parts of the diagram of the already constructed training workspace into the new workspace.

1. Select the part of the circuit diagram you wish to copy.
2. Press CTRL + C.
3. Switch to the page in the new workspace where you wish to paste the copy.
4. Press CTRL+ V. Paste the copy.
   When you work with function and location, all the component names obtain a dash "-".

Exercise 25-5: If several components must have a function/ location different from the page to which they belong, they could be assigned to another function/location using the Function/Location Box.

1.CA **Electrical**
2.CO **Function/Location Box** (**Function/Location** panel)
3.+ Fix the first point of the rectangle of the box.
4.+ Fix the second point of the rectangle of the box.
5.# B1
   Fill in the new value in the *Function* field.

6.# 02
   Fill in the new value in the *Location* field.
7.> **OK**
   The input box closes.
8.> **Yes**

   Confirm the renaming of all the components inside the Function/Location box.

   The new identifications have been assigned to all components located within the drawn rectangle.
   In the "Workspace properties" there is a setting that allows you to define if the function/location boxes will change the names of the cable cores or slaves found inside the box too.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercise 25-6: If the name of the Function/Location box has to be changed, proceed as follows:

1.+ Double-click the Function/Location box whose texts you want to change
2. Type in the correct names in the corresponding fields.
3.> **OK**
The input box closes.
4.> **Yes**
Confirm the renaming of all the components inside the Function/Location box.
All the components inside the Function/Location box are renamed.

The size of a function / location box can be changed with the help of the trackers. The size of the texts belonging to the box is not changed.

If, after a modifiction, a component is not located inside the box any more, it will show its function / location texts. If a component is found inside box after modification that has not been inside before, it will also show its function/location texts.

Please, disable trackers directly when you do not need them.

Exercise 25-7: If some separate components must have another function/location, they can be assigned to the function/location as follows:

1.+ Double-click the component whose function and location you wish to change.
2. Fill in the correct values in the appropriate fields.
3.> **OK**
Close the dialogue box.

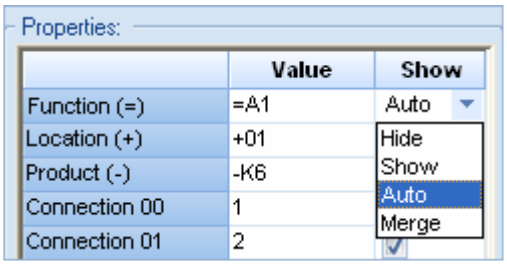
Exercise 25-8: View the Products list and the Documents list.

1. Open the Graphical Lists area in the Workspace Explorer.
2. Select Documents.
3.M Right-click and execute the **Generate** pop-up command.
4. Double-click the 0001 page to open it.
5. Repeat the same procedure for the Products list.
The components are displayed with their different functions/ locations.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

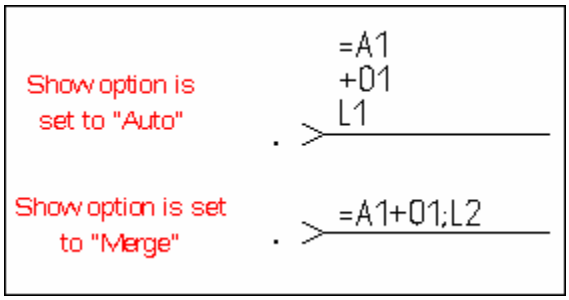
Hint

If you use function/location, it is possible to set the Show option for function/location in three levels by editing the component: ***Auto*** *(the function/location is displayed only if it does not correspond to the values set for the page),* ***Hide*** *(they are never shown),* ***Show*** *(they are always shown). Furthermore, it is possible to set the* ***Merge*** *option to show the function, location and product name in one line.*



The Merge setting is helpful especially for potentials when their Function/Location is different from the page where they are located.

As you can see in the illustration below, setting the ***Show*** *option to* ***Merge*** *allows displaying the texts in one line. In this way, potentials can be placed as usual with a distance of 5 mm between each other.*



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

BB REVISION MANAGEMENT

(Standard)

The revision management system allows you to update automatically any page on which data has been modified including cases when the change is a result of an automatic *SEE Electrical* functionality. *SEE Electrical* automatically inserts the new date on the corresponding pages.

To activate this function, tick the "***Change Revision Date Automatically***" option within the ***General*** tab of the ***Workspace Properties*** window.

This feature is particularly useful in the case of relay coils and their contacts, typically represented on more than one page.

For example, you move one contact in page 12 (with the relay coil in page 7) and save the workspace afterwards. *SEE Electrical* will automatically change the revision date on page 12 as well as on page 7.

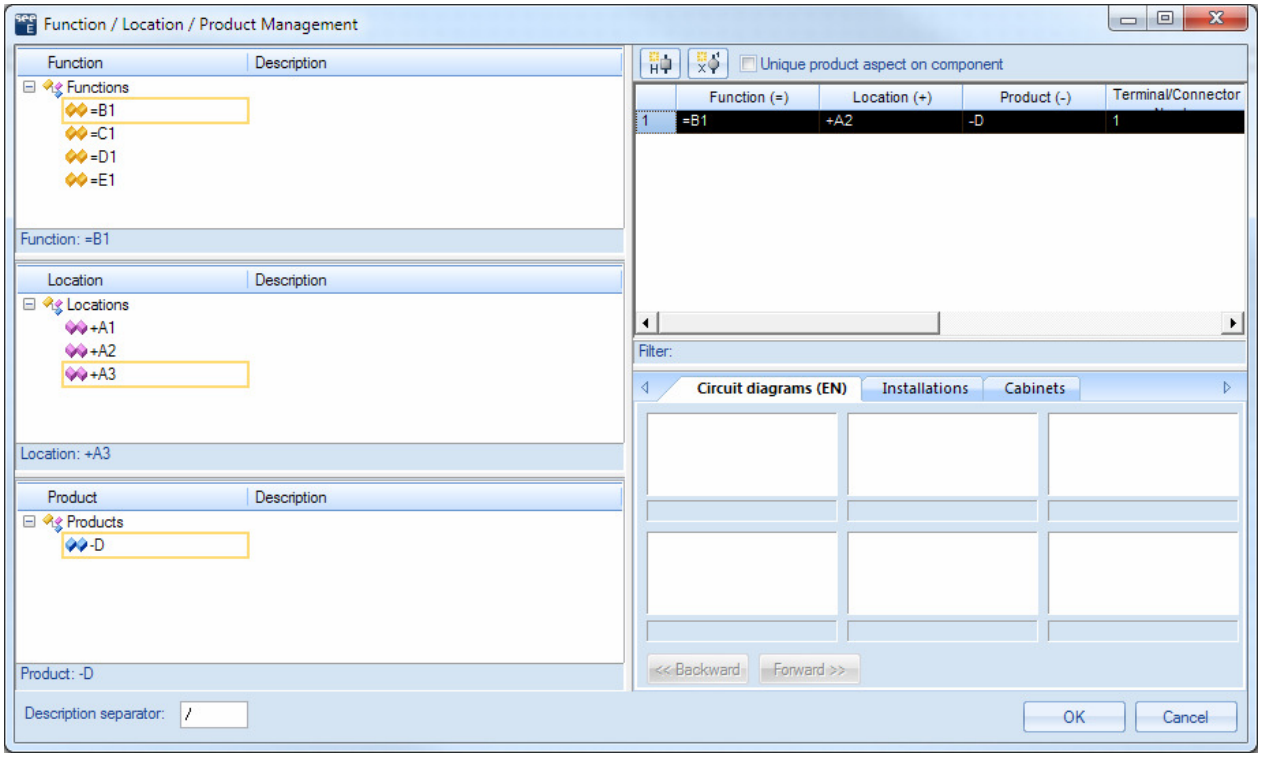
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

CC ADVANCED FUNCTION/LOCATION AND PRODUCT MANAGEMENT

(Advanced)

CC.1.DATABASE FOR FUNCTION/LOCATION AND PRODUCT MANAGEMENT

The **Aspects** command from the ***Commands*** tab allows you to define and manage functions and locations as well as the product database for the current workspace.
This command allows you to define the functions/locations and the product database for the whole workspace before you start drawing or to add them later. You can also add some descriptions, as well as display the pages and/or components which belong to a function/location.

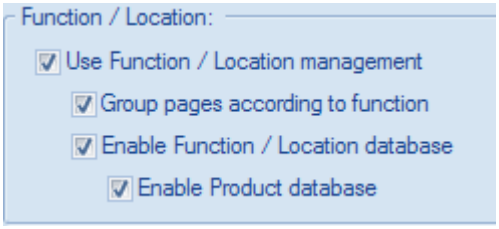


Notes

1. *When you launch the command for the first time, in case some functions/locations already exist in the workspace, they will be displayed. Otherwise, the window appears empty.*

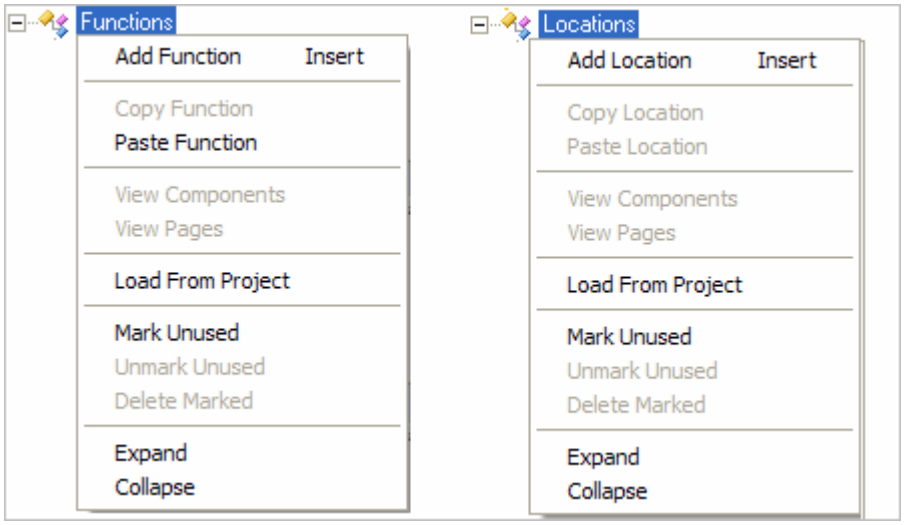
*2. If you use the **Aspects** command from the **Commands** explorer without activating the settings for function/location management in the Workspace Properties window, the "Use function/location management" and "Group pages according to function/location" options will be automatically activated. In case you do now wish to group the pages according to their function/location, you need to deactivate this option manually afterwards.*

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

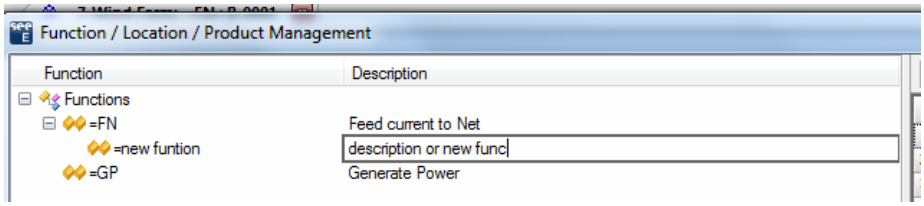

Function / Location:
Use Function / Location management
Group pages according to function
Enable Function / Location database
Enable Product database

To add a Function\Location:

1. Right-click to display the pop-up menu for **Functions** or **Locations**.



2. Select the **Add Function** or the **Add Location** pop-up commands to create new functions/locations.
   You can create functions/locations hierarchically, by selecting an existing function/location and executing the **Add Function** or **Add Location** pop-up commands.
3. Type in a name and a description (optionally) directly in the field that has been created.



To modify a Function\Location:

1. Select the function or location you wish to modify.
2. Type in the new values in the appropriate field.

To delete a Function\Location:

1. Select the function or location you wish to delete.
2. Right-click and select the **Delete Function** or **Delete Location** pop-up command.
3. Click **Yes** to confirm the deletion.

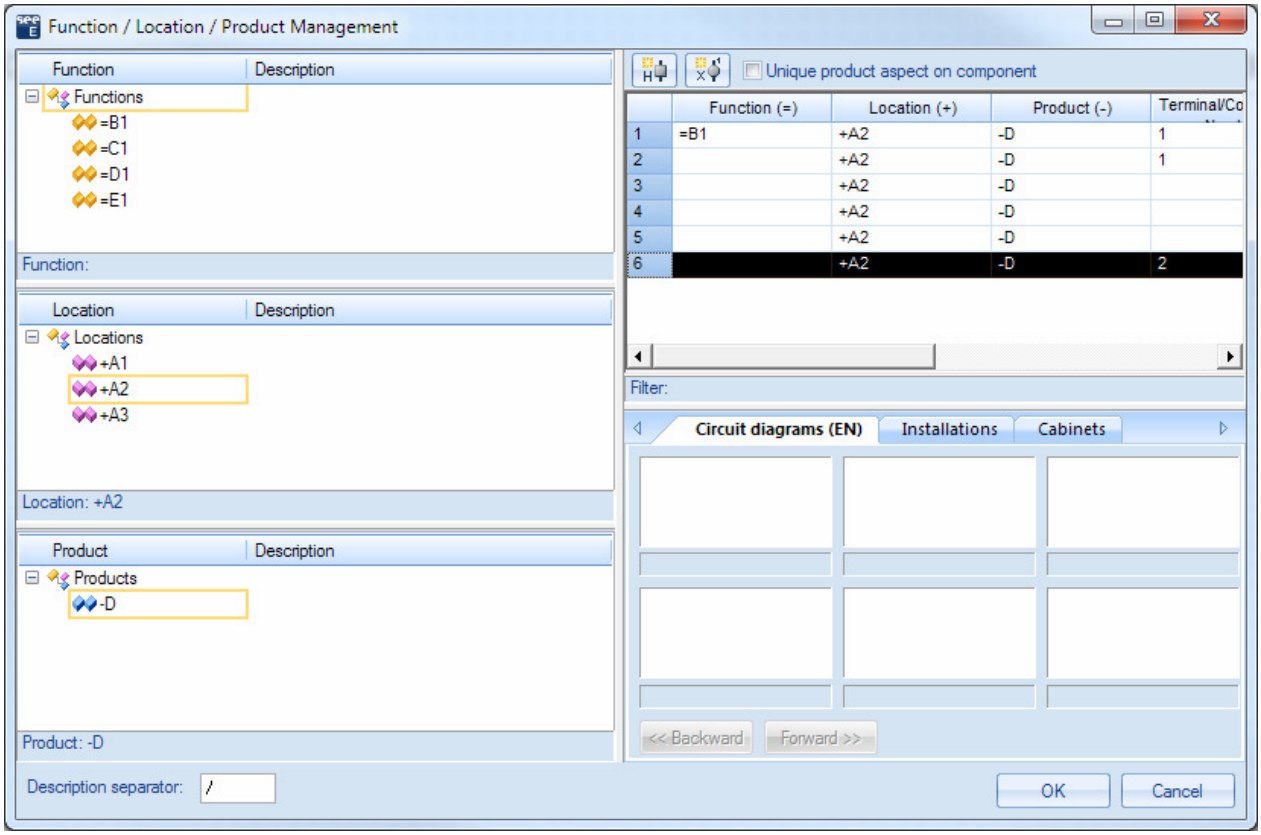
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Note

If there are pages or components belonging to this Function/Location, a warning message appears and the deletion is cancelled.

To display components belonging to a Function\Location:

1. Select a Function or a Location.
2. Right-click and select the **View Components** pop-up command.
   The components belonging to the respective function/location appear in the right pane of the window.



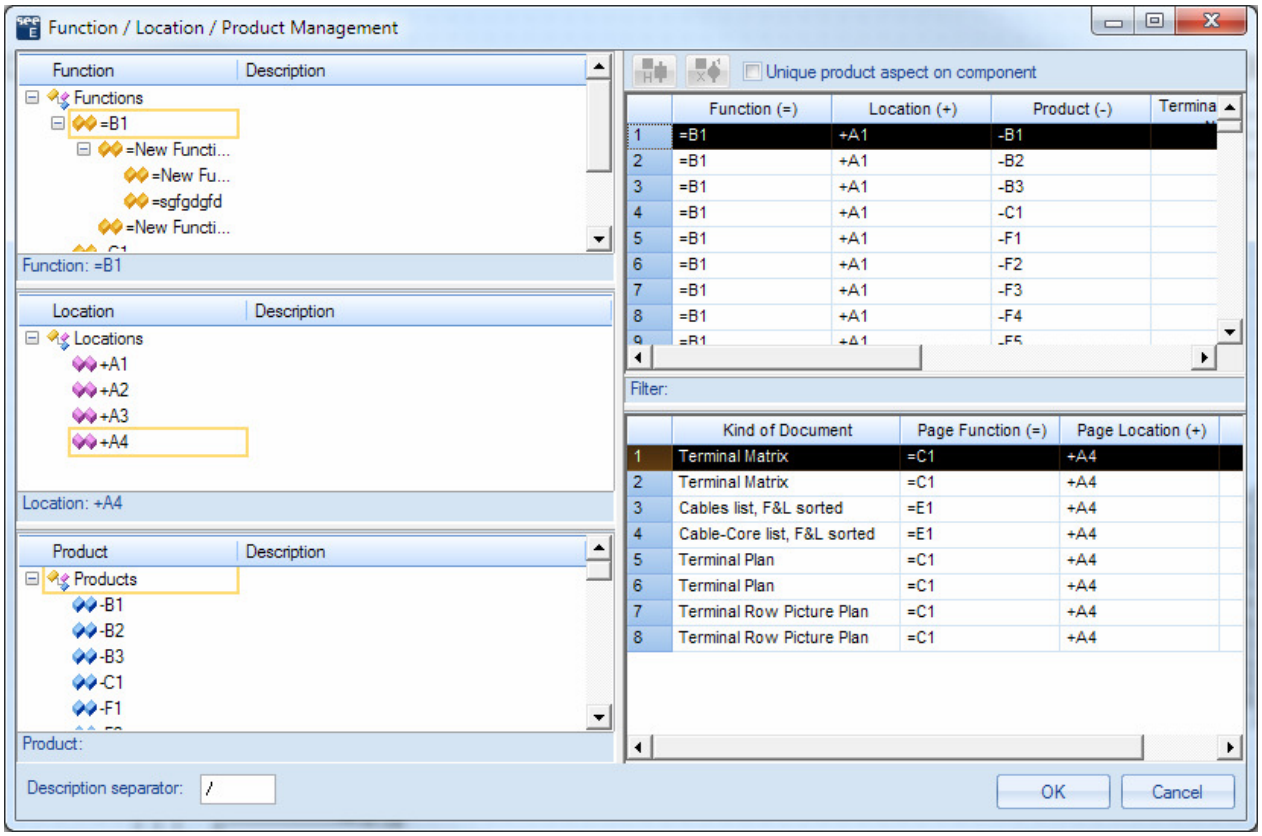
If you double-click a component, *SEE Electrical* opens the page where it is located.

To display pages belonging to a Function\Location:

1. Select a Function or a Location.
2. Right-click and select the **View Pages** pop-up command.

   The pages belonging to the respective function/location appear in the right pane of the window.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



If you double-click on a page, *SEE Electrical* opens it.

To load Function\Location information from project:

If the Function/Location information from the project is not read properly when you open the Function / Location Manager for the first time, you can use this command to recover the missing data.

To collapse / expand the Function/Location tree:

If you click a function/location and you execute the respective pop-up command, you can expand/collapse the tree structure.

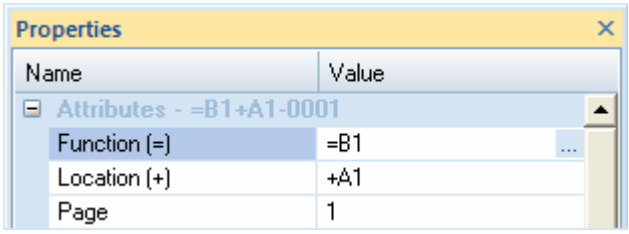
To use the Function/Location Database in a circuit diagram:

In case the "***Enable Function/Location database***" option is active in the ***Workspace Properties*** window, you can access the Function/Location Management window and select the desired function/location for the pages and components in your workspace.

To select a function/location for a page:

1. Execute the **Home** □ **Information** □ **Page** command and click the ... button in the "***Function***" field.

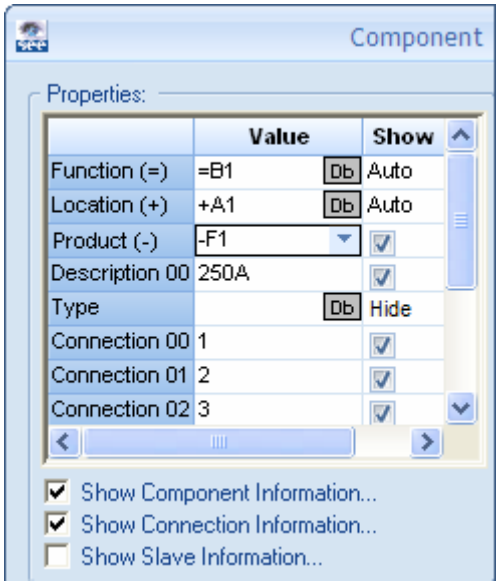
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



The ***Function/Location Management*** window appears, allowing you to select a function and/or a location for the page.

To select a function/location for a component:

1. Double-click the component to open the ***Component Properties*** window, for example:



2. Click the Db button in the "**Function**" field.
The ***Function/Location Management*** window appears, allowing you to select a function and/or a location for the component.

To Export parts of the Aspects tree

You can export parts of the Aspects tree into an XML file and later import then into a new (or empty) project.

1. Select the node you want to export.
Only the information from the selected node and the structure below this node will be exported. If you want to export information for the function and location, you have to export to 2 XML files.
2. Select the **Export To XML** pop-up command.
3. Define the name of the XML file.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

To import function/location information into the Aspects tree

You can import function/location information into the Aspects tree from an XML file.

1 Select the node you want to import the information into.
2. Right-click and select the **Import from XML** pop-up command.
3. Decide if you want to overwrite the existing information.

Note

The functions and locations can also be managed via the Function/Location database editor.

To add products:

If you have enabled the Product database in the ***Workspace Properties*** window ***General*** tab, the Products area is available in the ***Function/Location/Product Manager***.

1. Right-click within the ***Products*** area
2. Select the **Add Product** pop-up command.

You can add information about the new product in the new entry for a Product that appears. Within this line it is possible to insert a product's name and description.
If you define a component code, you can use the **Renumber** command to renumber the components later on, if necessary

To add components

The **Add Component** command is also available in the right pane of the window - . You can also assign a type to the component in the right pane.**Add Terminal/Connector** (Hotkey SHIFT + ->)- adds a terminal/connector to the selected product. This command is also available in the right pane of the window - .

After you execute the command, a window appears where you can define the number and sorting of the terminal/connector. Define how many terminals or connector pins will be generated in the "***Amount***" field. In "***Offset***" field, you can define the spacing between the values in "Terminal/Connector sorting".

To copy, paste and delete products:

Use the available pop-up commands to execute the different actions.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

How to avoid inserting components with the same product aspect

The "***Unique Product aspect on component***" The functionality is available in the right part of the window only in case a circuit diagram page has been opened. It is used to avoid inserting components with the same product aspect twice.

You can use the **Mark unused** command to find which products are not used.
In case you want to use one component at two different places, disable the setting and place the component a second time.
If the functionality is enabled and a product is already present in a workspace, the **OK** button is greyed out in case you select this product (-) aspect again.

How to use the Product Aspect

When working with the Product aspect, component (and cable) names have to be assigned with the help of the *Function/Location/Product manager*. (the component attributes "*Description00*" and "*Type*" are filled from the "*Component*" definition here.) The function and location currently chosen in the *Function/Location/Product manager* when making a component in the "product" area are used in the component name.

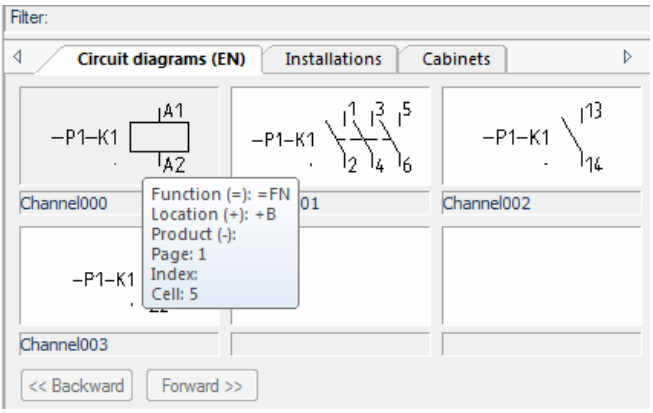
After a component is created, a type can be added from type database.
You can also add terminals and connectors with the appropriate command.
If components are defined in the Function/Location/Product manager and the type used has a proper channel definition (connection texts and symbol to use in circuit diagram are defined), you can double-click the graphical symbol in the lower right area of the window and place it directly into an open diagram.
If a component contains more than one channel, all available channels are displayed in this section. The channels that have been already used are greyed out. On mouse over they show the page and cell where the symbol is located. In this case you can navigate to this page by double-clicking in case the window has been opened via the **Aspects** command.

How to change component names, when working with product database:
If you want to change the name of a component, you should select the name in the ***Components*** area, select the correct aspects for the component and apply them with the **Change aspect** command. (If the ***"Unique product aspect on component"*** option is enabled, no component with the chosen product aspect can exist. If it does, you have to delete it first.) If you use the **Change aspects** command, all appearances of the component in workspace (for example linked slaves etc.) also change their names.
If you just select another name for a component that has several appearances in the workspace, the names of the other appearances are not changed.
If you want to attach a slave to another master, you have to select the new name.

Types with channel definition - the Symbol Explorer section

If a channel definition is properly defined for a type used on a component, in the lower right part of the aspects manager window are displayed the symbols defined in the channel.
You can double click a symbol in this symbol explorer section and place it in the current drawing.
After a symbol is placed, it is greyed-out and the page and cell it is found in are displayed on mouse over.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



By double clicking on an existing (= greyed-out) symbol in the symbol explorer section, you will be navigated to the page where this part of the component is used.
If symbols for different modules are defined in the channel, all are found in the tab referring to the module.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

CC.2.USING NESTED ASPECTS

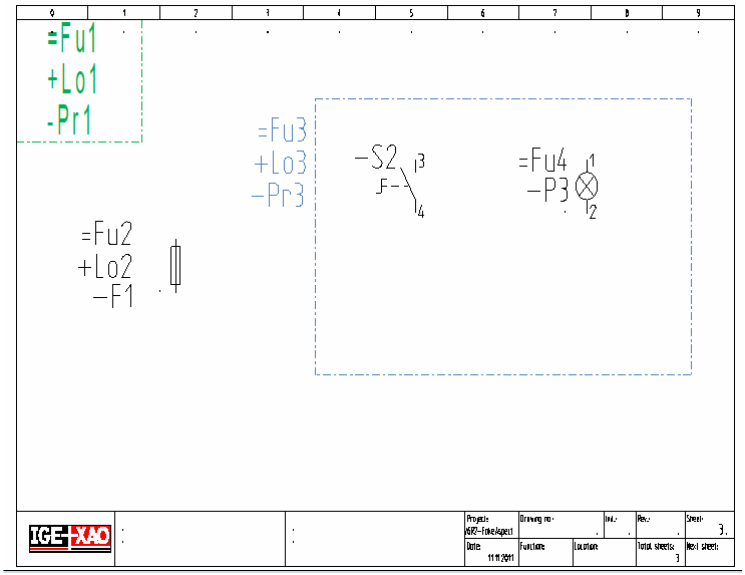
(Advanced)

An option within the ***Workspace Properties*** window offers you support of relative and absolute aspects (nested aspects). The option can be enabled if no pages exist in the workspace.

First Example for the use of nested aspects

The aspects from the page are added to the aspect found on each object. The box transfers its aspects to the components it contains.

Object Reference Designation
Page =Fu1 / +Lo1 / -Pr1
Fuse =Fu1=Fu2 / +Lo1+Lo2 / -Pr1-F1
Box =Fu1=Fu3 / +Lo1+Lo3 / -Pr1-Pr3
Switch =Fu1=Fu2 / +Lo1+Lo3 / -Pr1-Pr3-S2
Lamp =Fu1=Fu3=Fu4 /+Lo1+Lo3 / -Pr1-Pr3-P3
In this example all aspects seen at the components or the box are sub-aspects of the aspects defined for the page - they are relative aspects.



2nd Example for the use of nested aspects

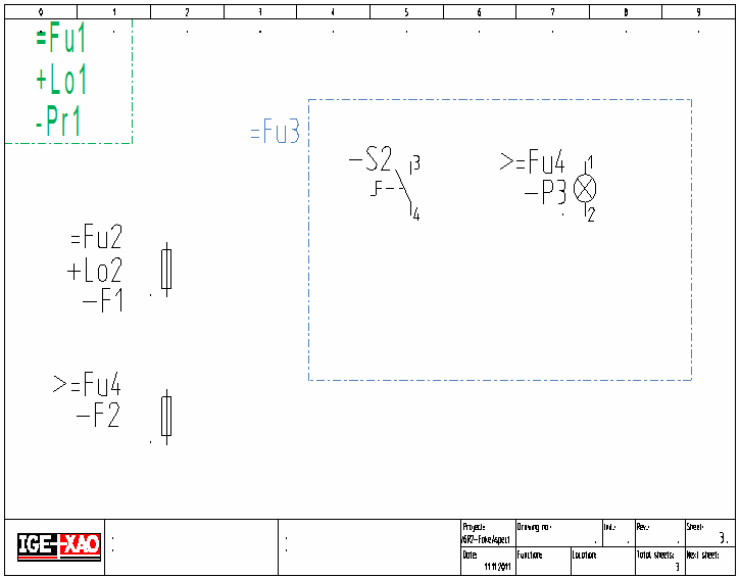
Only one aspect is used here, but the rule is valid for all three of them.
A workspace contains the following structure for the function:

= Fu1 (main function)
=Fu2 (sub function in Fu1)
=Fu3 (sub function in Fu1)
= Fu 4 (another main function)
The page has assigned function =Fu1

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

The fuse -F1 is relatively referenced, this means that it is found in the sub-function =Fu2 of the main function =Fu1.
The box and the switch are also relatively referenced.
Absolute references show a ">" in front:
The fuse -F2 is placed in the main function =Fu4 which is not a sub -unction of =Fu1. So the fuse -F2 is absolutely referenced, and that is why it has the > sign in front of the function text.

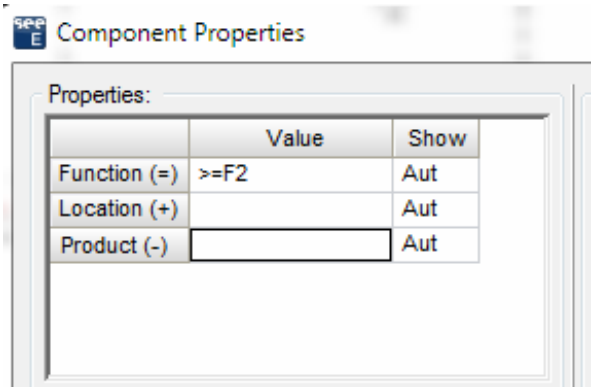
The lamp is also absolutely referenced to function =Fu4.
Object Reference Designation
Page =Fu1 / +Lo1 / -Pr1
Fuse -F1 =Fu1=Fu2 / +Lo1+Lo2 / -Pr1-F1
Fuse -F2 =Fu4 / +Lo1 / -Pr1-F2
Box =Fu1=Fu3 / +Lo1 / -Pr1
Switch =Fu1=Fu3 / +Lo1 / -Pr1-S2
Lamp =Fu4 /+Lo1 / -Pr1-P3



How to define absolute aspects on components

First Case: The Aspect database is not used

Add the ">" sign in front of the aspect you want to define an absolute value for.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Second Case: The Aspect database is used

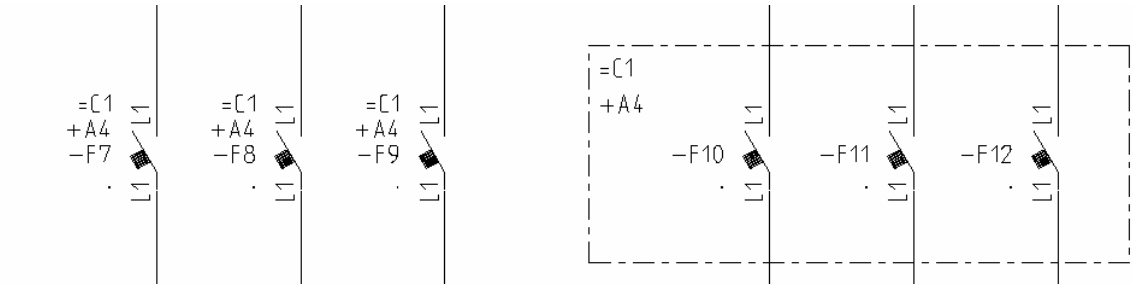
The aspects have to be defined in the ***Aspects Manager***.

If the component not found inside a function / location box and an aspect is chosen that is not a child of the one defined on the current page, the > is created in front of this aspect on the component

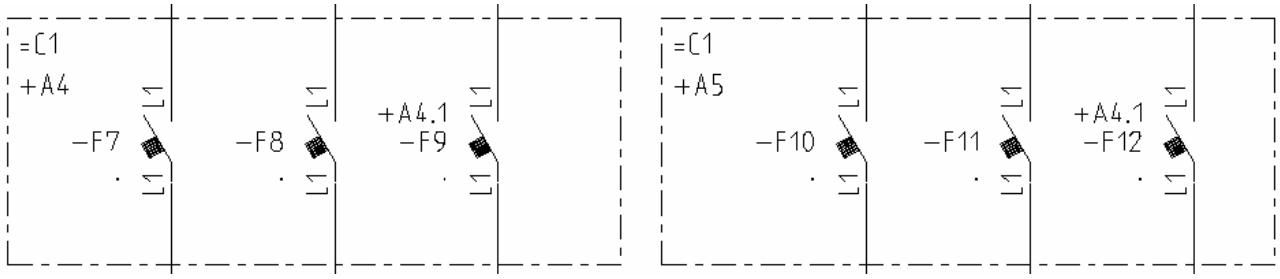
If the component is found inside a function / location box and an aspect is chosen that is not a child of the one defined for the function/location box, the > is created in front of this aspect on the component.

How to define relative aspects for function / location boxes.

If you use a function / location box to hide the aspects already defined for components you have to select **No** when the "***Rename all components inside this function/location box?***" question appears.



If the answer is **No**, the aspects for the components are not changed - only the view of the texts is updated.
If the answer is **Yes**, the aspects from the box are added to the ones already present on the components. The -F7 component will be =C1=C1+A4+A4-F7
If the aspect defined on a box is changed and changes are applied to the components inside the box, only the affected part is changed.



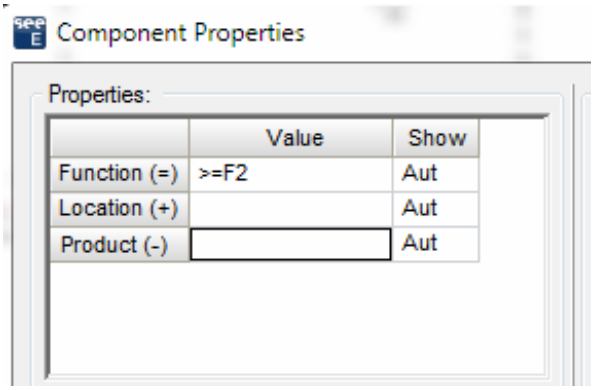
The location aspect for the -F7 component is +A4, the location aspect for the -F9 component is +A4+A4.1. After location is changed from +A4 to +A5, the location aspect for the --F7 component is +A5, the one for -F9 is +A5+A4.1 so only the affected part is change.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

How to define absolute aspects for function / location boxes

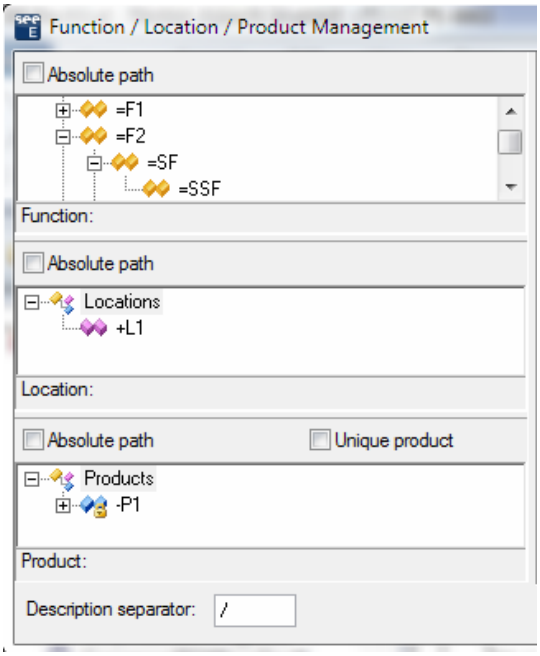
First Case: The Aspect database is not used

Add the ">" sign in front of the aspect you want to define an absolute value for.



Second Case: The Aspect database is used

The aspects have to be defined in the ***Aspects Manager***. When choosing the properties for a component or function/location box, the "***Absolute path***" option is available and can be activated for the absolute aspects.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

DD COMPLEX MODIFICATIONS OF THE DATABASE LISTS

(Advanced)

DD.1.1. PRODUCT EDITOR

You can edit one or several records. If a record is selected, you can change everything within this record in the right pane of the editor. If several records are selected in the list by pressing CTRL or SHIFT, you can change only description, type, and function/location, if needed. The changes apply to all of the selected components.

The **Renumber all components on all pages** function is available as a pop-up command and allows you to rename all the components on all the pages, as long as the user-defined Component Numbering setting is not activated (from *standard* level).

DD.1.2. TERMINAL EDITOR

You can edit one or more records.

The **Renumber all components on all pages** and **Renumber all shown terminals** (=filtered) functions are available as pop-up commands.

Edit multiple records:

The sequence of defining the sorting criteria is important. The criteria are shown in the headline of the window.

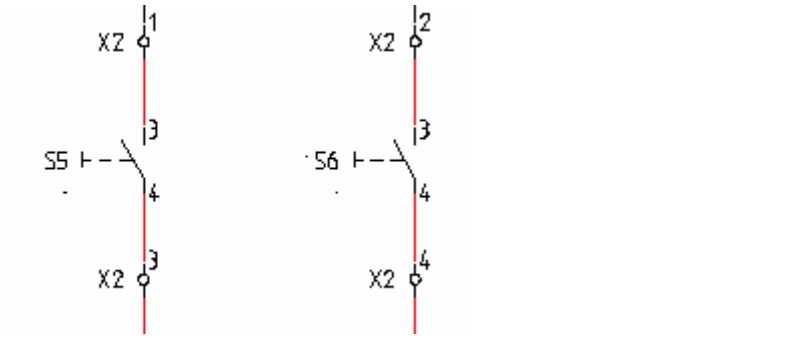
a) Change the terminal names for the whole terminal strip:
   - Select all terminals in the terminal strip. Change the terminal names.

b) Renumber all terminals in the terminal strip
   - Select all terminals in the terminal strip.

In the "***Terminal number***" field, enter the following: 1>1 (start from terminal number 1, increment 1)
or
1>10 (start from terminal number 1, the next terminal has an increment 10, therefore it is 11)
This method can also be used for terminal index.

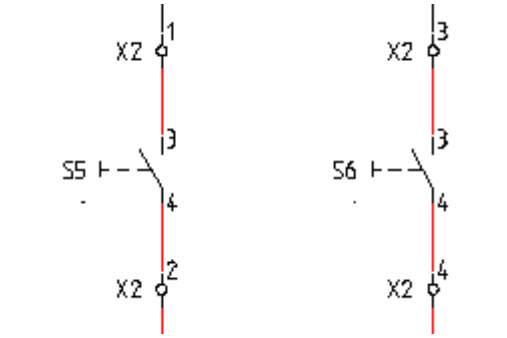
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

c) Renumber the terminals in a terminal strip as follows: all the terminals on page 1 in the top line first, then all the terminals below (starting in column1) etc.



The terminal editor sorts directly so renumbering in this way is possible. Select the desired terminal strip in the "***Terminal strip***" column**.** Right-click with the mouse and set the filter on. Right-click and select **Renumber all shown terminals** from the pop-up menu**.**

d) Renumber the terminals in a terminal strip not in the available sequence but all terminals in column 1of page 1 first, then the terminals in column 2 etc.



- Select the desired Terminal strip in the "***Terminal strip***" column.
- Right click and set the filter on.
- Select the "***Page***" column.
- Right-click and sort the column in ascending order.
- Select the column X – it is sorted in ascending order, and then the Y column is sorted in descending order. Right-click again and click **Renumber all shown terminals** from the pop-up menu**.**

e) Renumber all terminals, except terminals with number PE, etc.
- Sort by terminal strip, then by terminal numbers, select all the terminals except these with PE number etc. and change as described under b).

f) Change the terminal type
- Select all the desired terminals and enter a type.

g) Renumber combined terminals in the ***Terminal Editor*** (*advanced level*)
In case you have multi-layer terminals in your circuit diagram, make sure to use the ">=" formula when you rename them in order to avoid omitting numbers during the renumbering.
For example: 1>=1.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

When you rename multi-layer terminals, where there is more than one terminal with the same name, number and sorting, a message appears, asking you whether to rename all components. Click the **Rename All** button to make sure all single levels of multi-layer terminals remain combined.

DD.1.3. CABLE EDITOR

Multiple records can be edited at the same time. By selecting the **Check cables** pop-up command, you can check out overfilled cable-cores.

Cables can be automatically renumbered if a component code is defined for them in the ***Cables*** tab of the ***Circuit Diagram Properties*** window and if an automatism for renumbering of components is enabled.
The renumbering is possible in the "Editor, cable" database list via the **Renumber all cables on all pages** command.

DD.1.4. CONNECTOR EDITOR

The "Editor, Connector" allows you to change the information for the connectors and their pins. Multiple records can be edited at the same time.

In the editor you can change the name and the type of the connectors as well as the pin-ID. The name of the pin cannot be changed here, it is defined in the channel definition of the type. If a type is assigned to a connector and the pin ID is changed, the pin name is updated directly from the type information.

Pin Ids or pin names can be changed only if one pin is selected in the editor.

DD.1.5. SIGNAL EDITOR

Multiple records can be edited at the same time.

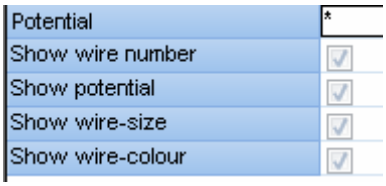
DD.1.6. WIRE EDITOR

Multiple records can be edited at the same time.

It is possible to switch the visibility for the wire texts in the Wire Editor. Depends on the settings in the wire properties (***Circuit Diagram Properties*** → ***Wires*** tab) if the change will be executed per wire and not per net.

If you select more than one entry, the visibility checkbox for the texts is dimmed, because in this case the different wires can have different options for the visibility.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



To switch the visibility in this case just click once to turn the visibility on for all the selected wires. If you want to turn it off, click a second time. To confirm the change, click elsewhere in the editor.

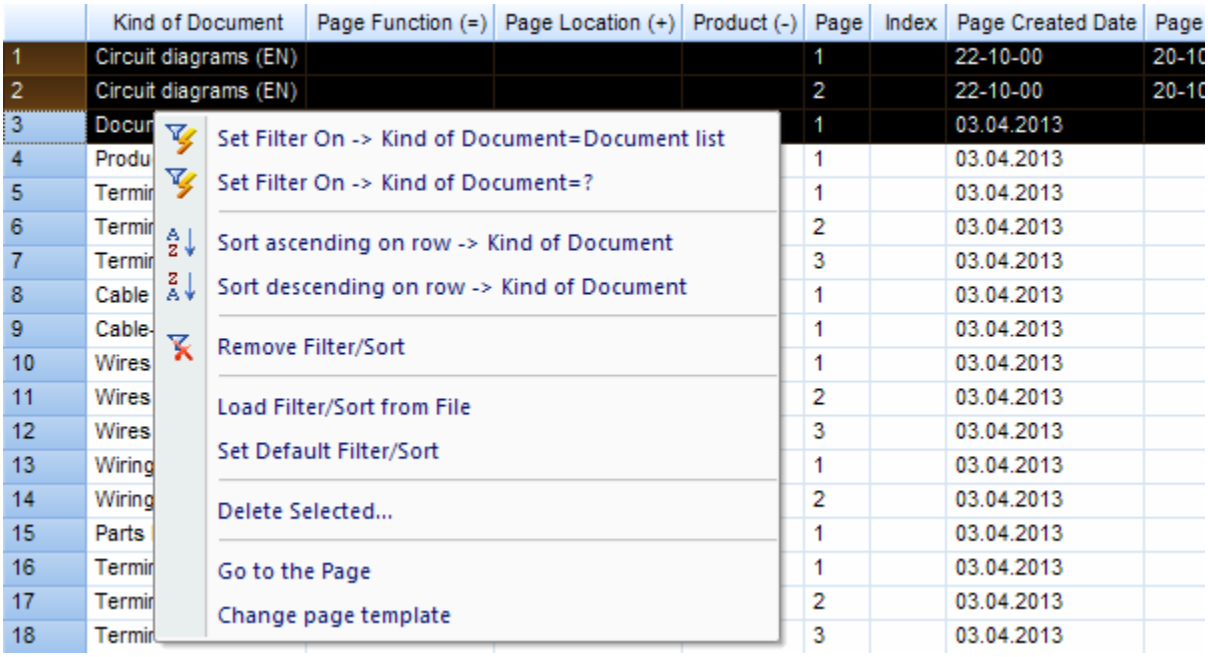
DD.1.7. FUNCTION/LOCATION EDITOR

Multiple records can be edited at the same time. The position of the page in the Workspace Explorer can be adjusted.

DD.1.8. DOCUMENT EDITOR

You can select, within this editor, multiple documents from a project and delete them in a single operation by using the **Delete Selected…** pop-up command.

In the "Editor, Document" you have the possibility to change the page template for the selected pages.



- Select the pages you want to change the template for.
- Right-click and select the **Change page template** pop-up command.

If you have selected only one document, a message appears asking you if you want to clear the entire page, or if you want to change only the template.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

If you have selected multiple documents, a message appears asking you if you want to clear the drawing (or change the template) only on the active page, or on all of the selected pages.

If you work with IEEE circuit diagrams, make sure that the number and the position of the sections are equal in the old and in the new template.

If you select different kind of documents, a message appears informing you that it is not possible to change the template unless selecting documents of the same kind.

DD.1.9. REDLINING OBJECTS LIST

Callouts are shown in the "View, Redlining" list with the text they contain and the page where they are found.

Because of this, callouts can be used to keep an overview about the revisions made in a workspace. The "View, redlining" list gives you the possibility to navigate to the redlining in the page via the pop-up menu.

If the database list is already open and you need to refresh it, please close and reopen it.

If the database list "View, Redlining" is opened, there is a question, if the existing "Redlining list" shall be opened or if a new one must be created.

This database list can then be inserted into a page using the **List2drw** command or in the ***Advanced*** level, a user defined graphical list can be created. (#Where Export_4212)

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

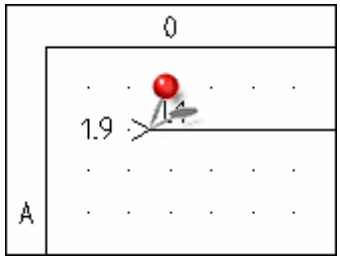
EE ADVANCED PROCESSING OF A PROJECT

(Advanced)

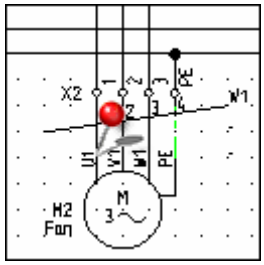
EE.1. NAVIGATION IN THE PROJECT

Two ways of navigation exist in *SEE Electrical*:

✓ Navigation via cross-references:
This method of navigation allows you to go from one page to another with the help of the cross-references between different components, for example potentials.
Double-click the cross-reference symbol on the potential L1 on page 1 of your training workspace.
SEE Electrical automatically opens page 2, where the source reference of the potential L1 is marked with a red pin.



✓ Navigation via the database editors
This method of navigation allows you to go from a component in the database editor to the page where it is inserted.
Open a database editor, for example the Product Editor.
Select a component, for example the motor 1M2.
Right-click and execute the **Go to the Page** pop-up command.
SEE Electrical automatically opens the page, where the component is located and it is marked with a red pin.



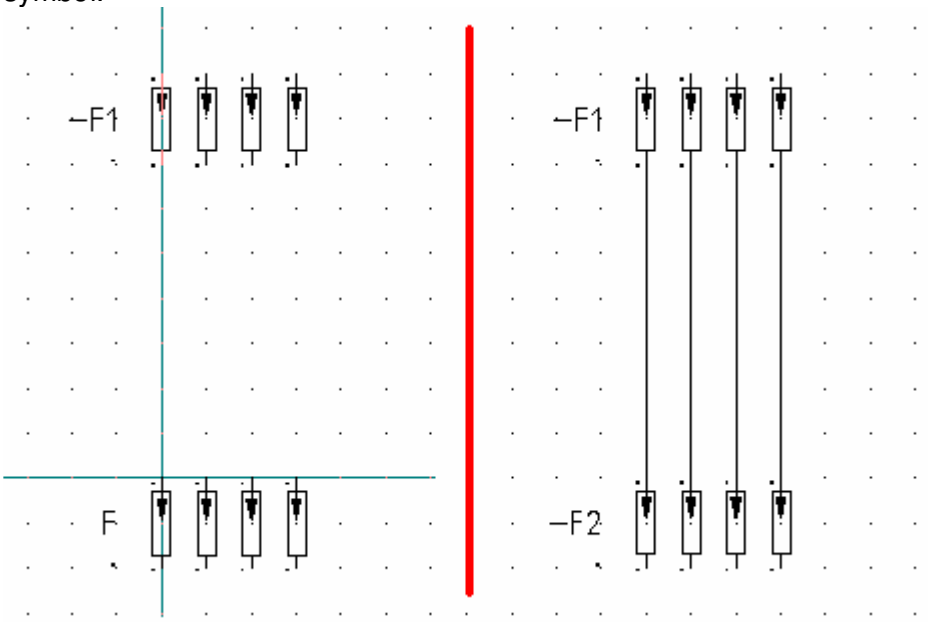
Navigation is implemented for some graphical lists such as Products list, Terminal list, PLC I/O list, Cable list, Products Assembly.
Tick the product name and select **GoTo** pop-up command.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

EE.2. WORKING WITH AUTOMATICALLY GENERATED CONNECTIONS

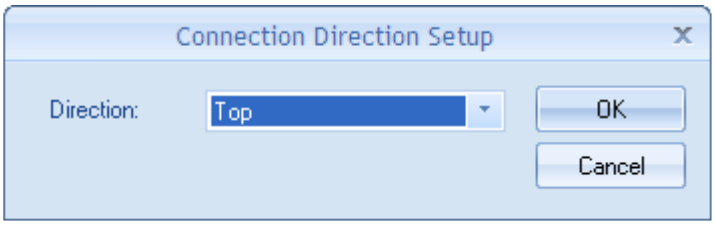
When adding symbols you can view the relevant wires by means of the **Electrical □ Wire Connections □ Auto Connection** command.

If enabled, the command automatically adds vertical/horizontal wires when new symbols are added. The example below shows screen shots (of the graphical representation) before and after adding a symbol:



When using the automatic wiring tool, you are enabled to define the route direction of the connection pin by the **Electrical □ Connections □ Direction** command.
By analyzing the shape of a given symbol, *SEE Electrical* in most cases calculates the possible directions for connecting the symbol (up, down, right, left). However, the shape of some special user-defined symbols may cause *SEE Electrical* to fail the calculation. In such case, *SEE Electrical* looks for the basic definition of directions set by the user.

The command is used to set this parameter as a default one. So, after activating it, specify the connection direction:



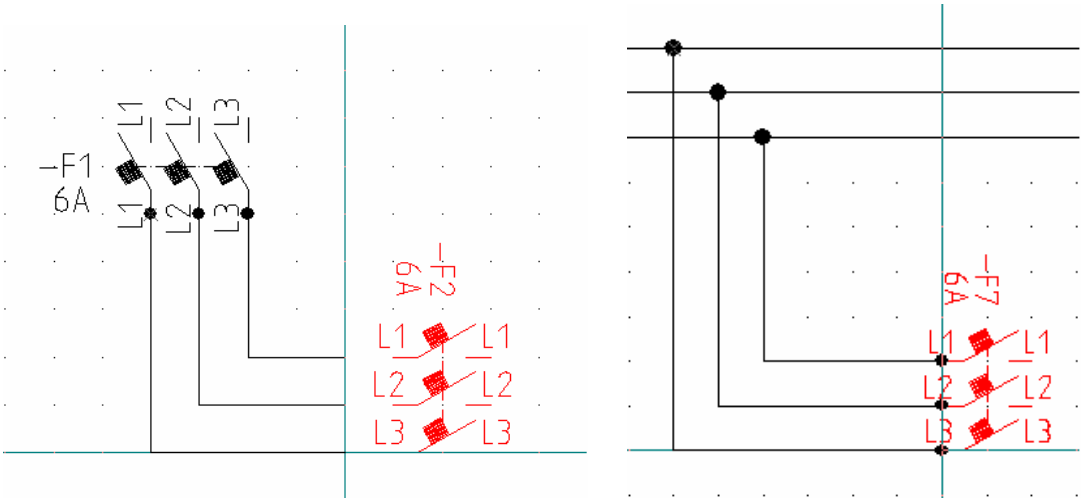
Important!

The command is only active and available when using the automatic wiring tool.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

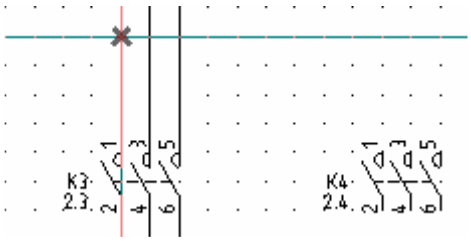
EE.3. ORTHOGONAL WIRING

The **Electrical** **Wire connections** **Orthogonal Wiring** command allows you to draw multi-mode wiring. You are allowed, optionally, to draw direct orthogonal wiring.



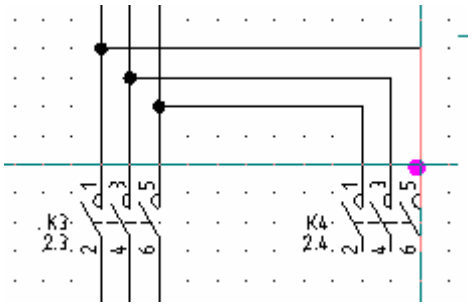
Exercise 29-1: Draw an orthogonal wiring. To do this, open page 1 of your workspace and delete all wires between the symbols which represent the motor reverse protection (columns 4 and 5). Proceed as follows:

1.CA **Electrical**
3.CO **Orthogonal Wiring** (**Wire Connections** panel)
3.+ Position the first point of the connection on the first top potential.
A wiring must always begin on an existing wire or on a component.
4.+ Move the cursor downwards to the motor.
5.+ Place the ending point of the wire on the connection point U1 of the motor.
Now continue to draw the missing wires between the second main contactor in the motor reverse protection. The **Orthogonal Wiring** command is still active.
6.+ Place the first point of the wire in the left vertical wire above the first main contactor.



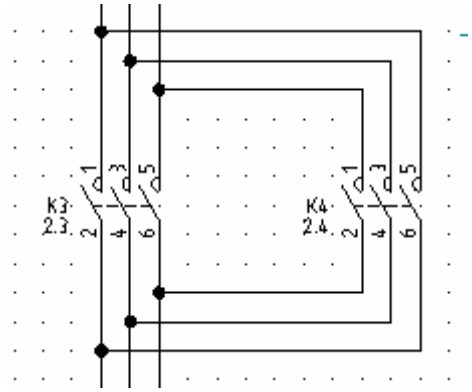
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

7.+ Move the cursor to the right until you reach the connection 5 of the second contactor.

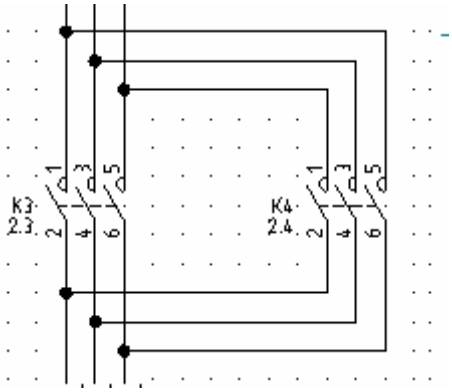


8.+ Move the cursor downwards and place a corner point under connection 6 of the second contactor. Then move the cursor horizontally until you reach the left vertical connection between the first contactor and the motor.

9.+ Click to position the ending point.
According to the cursor position, you will get the result shown on one of the two pictures below:



or

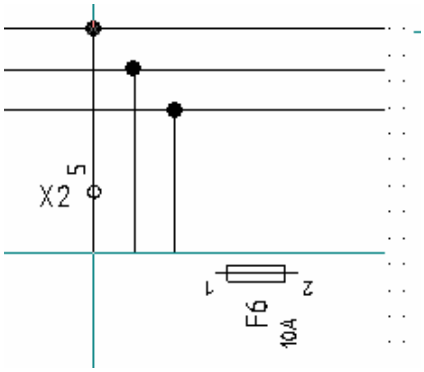


10.> Right-click to exit the orthogonal wiring mode.

Hints

1.By pressing the Tab key, you can toggle between horizontal and vertical drawing, provided that this is technically possible in the current situation.

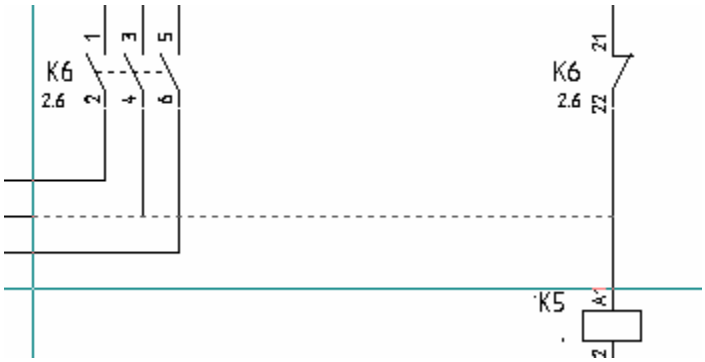
2. By using the Shift key, you can switch on or off the orthogonal creation of wires, if this is not appropriate in the current situation.



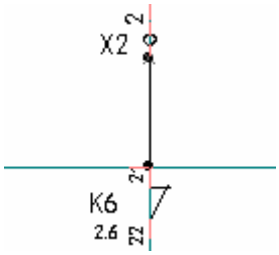
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

3. While drawing, SEE Electrical checks if the respective wire is possible. If this is not the case, the wire representation appears dashed.

4.



4. While you are drawing using this functionality, the connection points of the components are also visible.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

EE.4. ADVANCED FUNCTIONALITY FOR CABLES

EE.4.1. DEFINE CABLE NAMES AUTOMATICALLY

It is possible to define a component code for the cables. If you have not defined a component code in the "***Component code for automatic cable numbering***" field (***Circuit Diagrams Properties*** window ***Cables*** tab), the cable names have to be assigned manually.

If you have defined a component code, the "Component Numbering" method (***Circuit Diagrams Properties*** window ***General*** tab) is applied for the cables.

EE.4.2. DISPLAY CABLE CORES A SECOND TIME

Select the cable you want to display for a second time, for example, after a cross reference. The **Copy Ghost** command allows you to make a copy of an existing cable core. With the **Paste** command the copy can be placed elsewhere in the diagram. The copy is only "ghost" object. It cannot be edited. If you modify the original cable core, the changes are forwarded to the ghost copy. If the original cable core is deleted, a "?" appears on the copy of the cable core. It has to be deleted manually.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

EE.5. ADVANCED FUNCTIONALITY FOR POTENTIAL NUMBERING

With the **Define Signal Number** (**Electrical** category) command you can define the way of numbering of the signals for each connection. This is possible if you use the Signal type property.

If two components connected to one wire define different rules for the numbering, one is taken by random.

Make sure you do not set up a rule for the wire numbering that generates duplicated numbers.

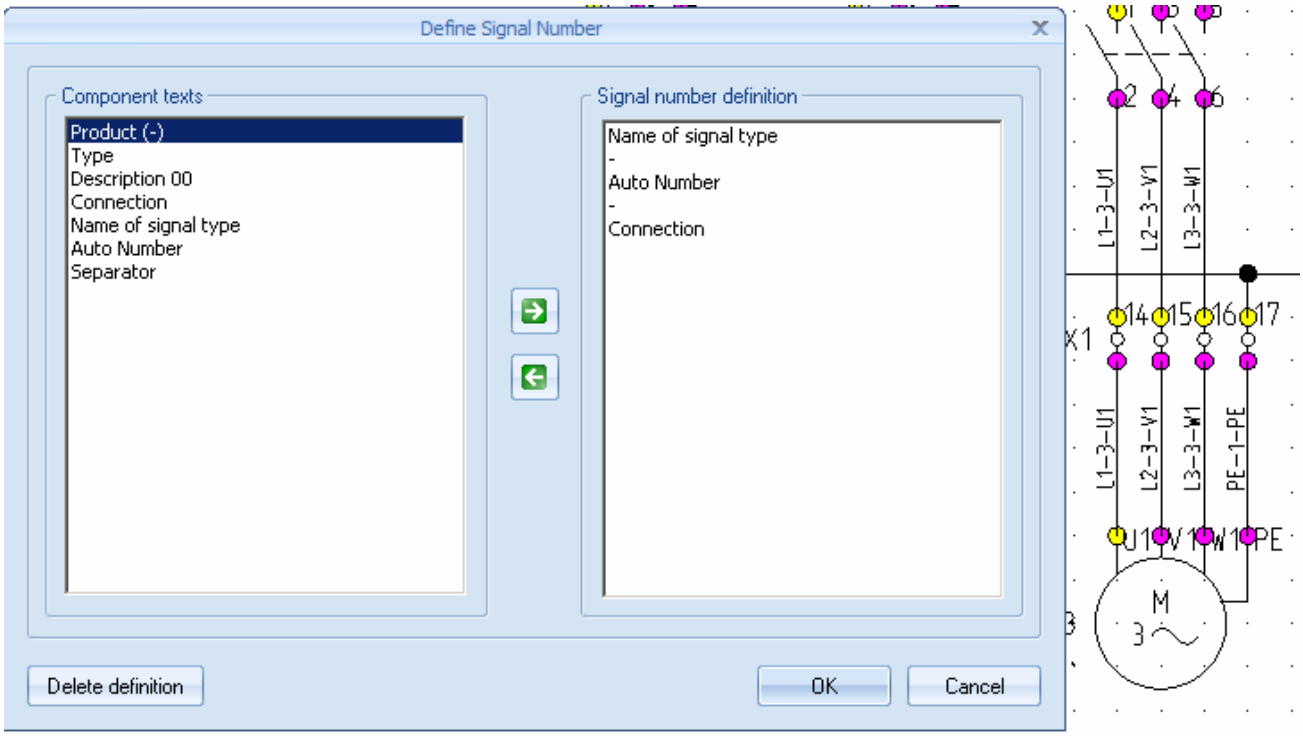
How to define a definition:

Select an entry from the ***Components texts*** area and move it to the ***Signal number definition*** area, using the button.

If you want to remove an entry from the ***Signal number definition***, use the button.

How to remove a definition:
Use the **Delete definition** button.

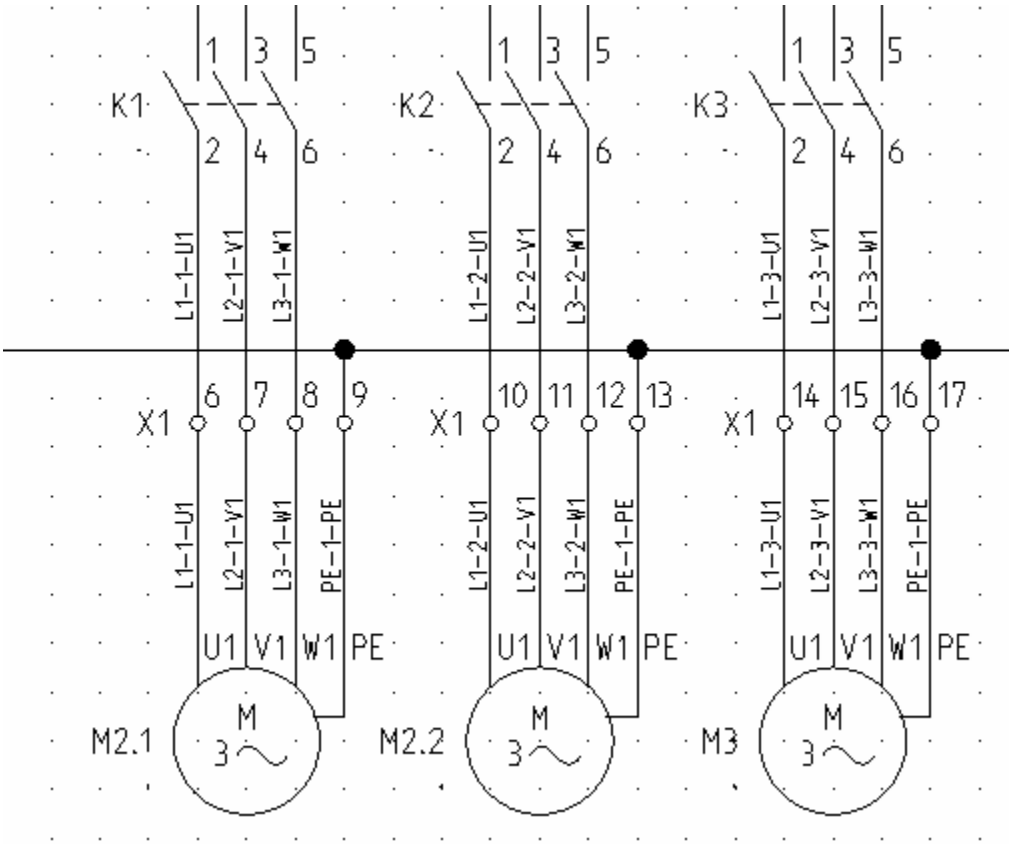
Example of a rule:



The separator can contain any text string, for example "-" or a text such as "Wire number".

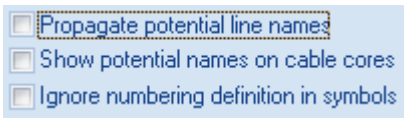
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Result:



The rule is stored with the symbol you defined it for.

If the "***Ignore numbering definition in symbols***" option inside the ***Signal setup*** dialogue (***Circuit diagrams Properties*** window ***Wires*** tab), the numbering definition from the connection is ignored and is used the definition from the signal type.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

EE.6. COPYING PAGES

EE.6.1. COPY SINGLE PAGE IN THE SAME WORKSPACE

If you select a page in the ***Workspace Explorer*** and you right-click with the mouse, a pop-up menu appears, allowing you to copy the current page. You can later paste it in the active workspace. When the command is activated, the ***Page information*** dialogue appears. It contains all page information texts existing in the copied page. Assign the new page number, modify any page information and click **OK**.

Component names are either automatically adapted, or you are asked to confirm their names if the "*Component Numbering*" is set to "*Free*". References are automatically updated.

The functionality is already available in the Standard level.

EE.6.2. COPY SINGLE PAGE IN THE SAME WORKSPACE OR IN DIFFERENT WORKSPACE

In the ***Workspace Explorer***, you can copy and/or move pages from one workspace to another using **Drag & Drop**. You can also do this within one and the same workspace.

Hint

To create a copy of the page and keep the existing one, press the "CTRL" key while moving the page.

The copying of pages between workspaces is only possible, provided that the following rules are kept:

- ✓ Both workspaces must be open.
- ✓ The module from which the page is copied and the module to which it is copied must be of the same type.
- ✓ For pages in the Circuit diagrams module, if Function and Location management is enabled in the source workspace:
  - ✓ Function and Location management must also be enabled in the destination workspace;

  or
  - ✓ If Function and Location are not enabled in the destination workspace, the page that is copied must have empty values for Function and Location.
- ✓ If the source Function is different than the destination Function, the copied page takes the value of the destination Function.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

The copied pages are exact copies of the original page. This means:

- ✓ If you copy a page within the same workspace, you have to change the names of copied terminals (in the diagram or via the ***Terminal Editor***), as they are duplicated to the ones on the original page.

  If you copy the page to a different workspace, this may not be necessary, since the terminals may not exist there before copying.

- ✓ If you copy a page within the same workspace, you have to change the names of copied cables (in the diagram or via the ***Cable Editor***), as they are duplicated to the ones on the original page.

  If you copy the page to a different workspace, this may not be necessary, since the cables may not exist there before copying.
  If you exchange the name of the cable on the copied page, only the cores of this cable receive the new cable name, as there is no link between the objects on the original page and their copies in the same workspace.

- ✓ For contacts and PLC I/Os, if the master is on the copied page, too, the link is still valid and the name of the master is inherited by the slave.

  If the master is on a different page, you have to assign the correct name. We recommend that you display the ***Component Properties*** dialogue and make sure the right name is assigned.

- ✓ The names of components are treated as defined by the selected component numbering method. This means, if "*Free*" is chosen for component numbering, you have to change the names, otherwise they are changed automatically.

Note :

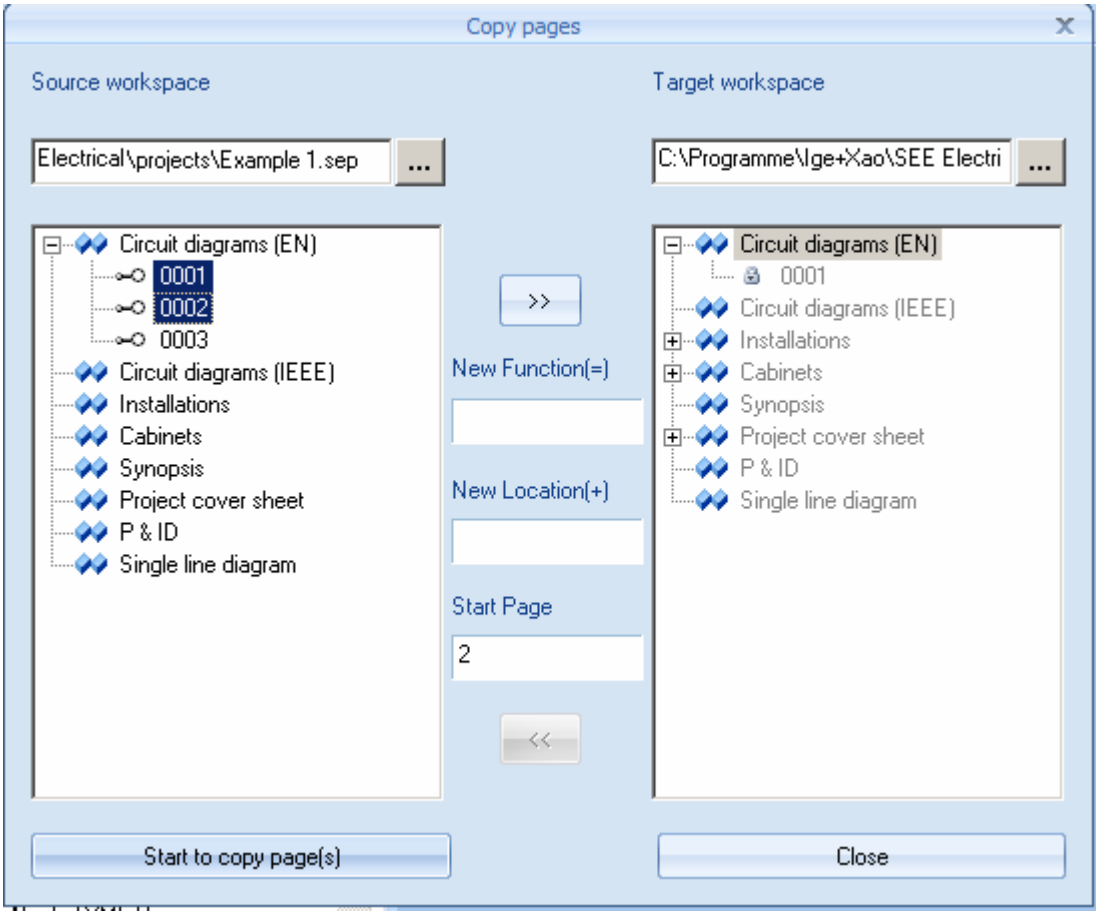
A message box appears informing you that only saved information will be copied. It also shows you the number the page will receive after copying. You can later change that number, if desired.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

EE.6.3.COPY MULTIPLE PAGES BETWEEN DIFFERENT WORKSPACES

In the ***Commands*** explorer, the **CopyP** command allows you to copy pages from one workspace to another.

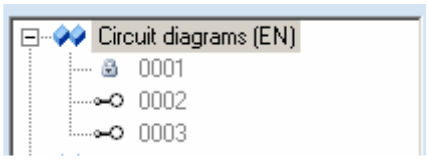
The target workspace has to be closed before executing the command.



1. Execute the command.
2. Define the source workspace.
3. Define the target workspace. The pages that exist in the target workspace are marked with a lock 0001 .
4. Insert a value in the "***Start Page***" field and, if necessary, information in the "***New Function (=)***" and "***New Location (+)***" fields.
   You have to make sure that the pages you want to copy do not exist in the target workspace. In this case, an error message appears.
   The selected pages will be inserted in a temporary target workspace, the number of the first new page will be the one you defined, the numbers of the other new pages will be consecutive to the one you defined for the first one.
5. Open the list of pages in the source workspace and mark the pages you want to copy.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

6. With the >> button move the selected page to the target workspace. .



If you make a mistake, you can mark the pages in the temporary target workspace and remove them by pressing the << button.

7. If you want to copy more pages, insert the new *Start page 2* again, select the pages to be copied and move them to the target workspace..
8. Use the **Start to copy page(s)** button to copy the pages to the target workspace.
9. Close the ***Copy pages*** window and open the target workspace.

If in the pages exist terminal strips or cables, their names will be changed. For example, from X1 and W2 to X1#0001 or W2#0001 to avoid duplicate names. You have to rename them manually or in the database editors.
The names of the slaves that are on the same page as their master are changed according the change of the name of the master. If the slaves are located on a different page, you have to control their names.

EE.7. CHANGING PAGE TEMPLATES

It is possible to change page templates for all circuit diagrams at the same time or for a single page:
The **Functions Change Page template on all pages…** command allows changing the page template in all of the circuit diagrams in a single operation.

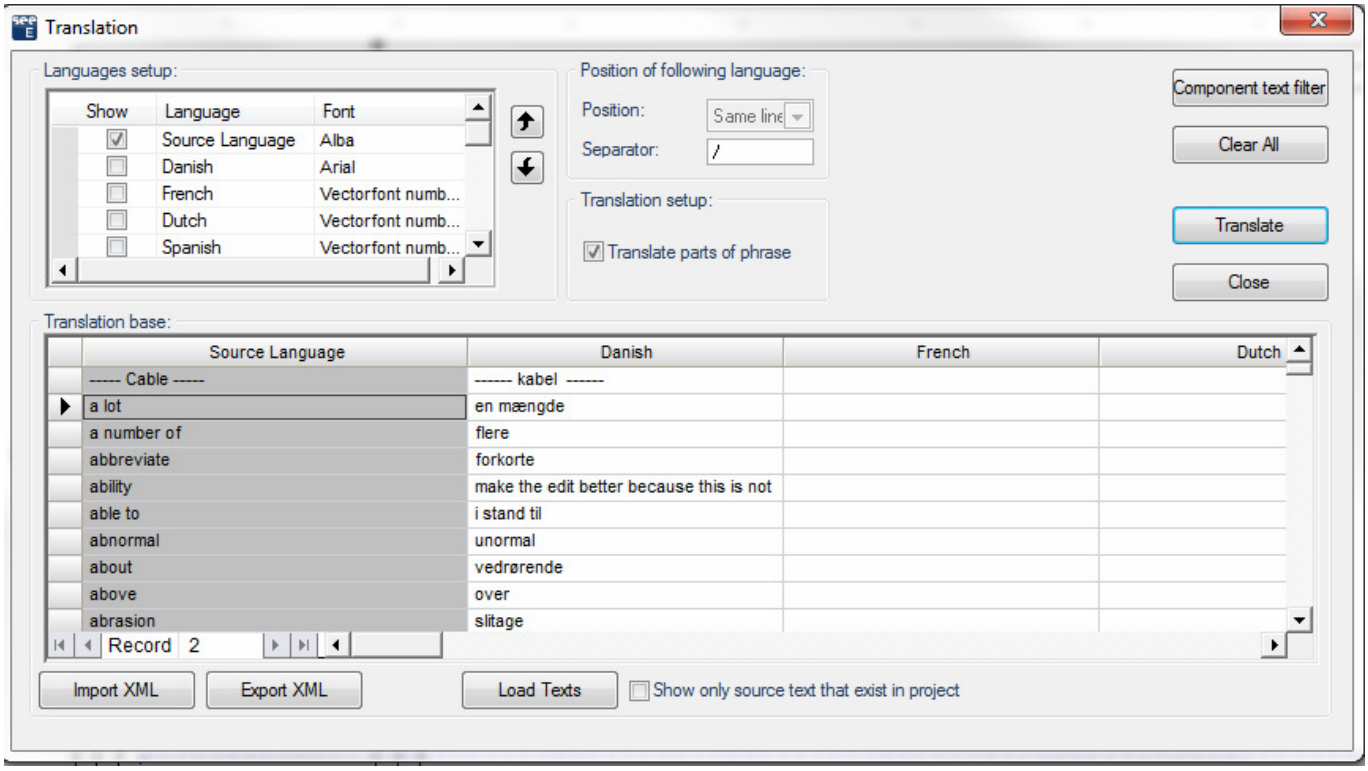
In the Document Editor you can select the pages for which you want to change the template and execute the **Change Page template on all pages** pop-up command.
The **File Open Page Template** command allows changing the page template for the current page. You can choose whether other objects to remain on the page or not.
In both cases, the page templates settings have an effect on the components, i.e. the component names may be customized.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

EE.8. TRANSLATING A WORKSPACE

(Advanced)

The **Functions** □ **Other** □ **Translate** command allows you to translate all texts contained in the circuit diagrams at the same time (consequently – all texts in the existing graphical lists, too).



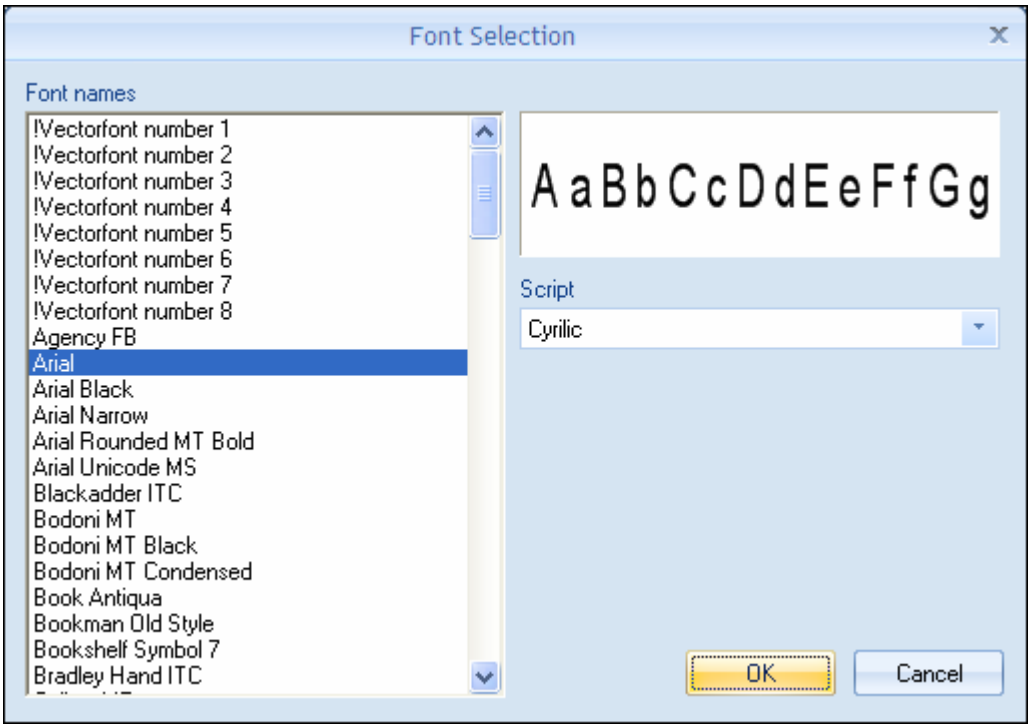
The source texts and the translated texts are stored in the *TRANSLATIONNEW.MDB* file (in the ... *\SYMBOLS* installation folder). The translation database can also be edited using *MS Access* or you can export it to XML, send the XML file to a translation office and reimport the XML file again.

You can also add your own texts to the translation database.

- ✓ If you wish to create a new line:
  - Select and existing line, right-click and choose the **Add New line** pop-up command.
- ✓ If you wish to delete entries:
  - Select the entry you wish to delete, right-click and execute the **Delete line** pop-up command. Please note that it is only possible to delete single lines.
- ✓ If you wish to create a second line in the entry:
  - Press CTRL and ENTER and continue with the input on the second line in the entry.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

✓ To display a translation with a different font proceed as follows: after clicking the ... button in the "***Font***" column for a language, you can select the specific font for your translations. If you are translating to languages that are not inside the range of the codepage you normally use, here you can change the settings for the font. In Western Europe, the script is Western, for Russian you have to use the Cyrillic script. Scripts are only available for Windows fonts, for example for "Arial".



The **Load texts** button allows you to load texts from the current workspace into the translation database.

The **Show only source texts that exist in project** button allows you to concentrate on the texts currently necessary.

Clicking on the **Component text filter** button opens a window in which you can set the texts to be translated (or not) selecting them by their IDs.

Attention!

*The **Edit** □ **Text** □ **Edit Text** command shows texts always in the source language. If the translatable text must be changed, change the source text if it is incorrect, or change the text in the target language in the translation database. Then run the translation process again.*

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

EE.8.1. DISPLAYING SEVERAL TRANSLATIONS AT THE SAME TIME

(all levels and Viewer)

It is possible:

- ✓ To show different translations in one line/in different lines:
  If you wish to display them in one line, you can choose a separator in the respective field.
  Languages can be moved to the desired position in the *Languages setup* area of the ***Translation*** window or via the **View** □ **Other** □**Language** command.

EE.8.2. TRANSLATION OF PART STRINGS

It is possible to translate part strings. The software searches the part strings in the following way:
First is searched the whole text to be translated. If a match has been found, the text is translated.
If the text is not found in the translation database, the software searches for a part string of the text.
The software searches first for the part string that has the largest number of characters. If this part string is not found in the translation database, the next one in the text is taken into consideration, etc.

Examples:

Source text	*French translation*
Motor	*Moteur*
Motor control	*Controle du moteur*

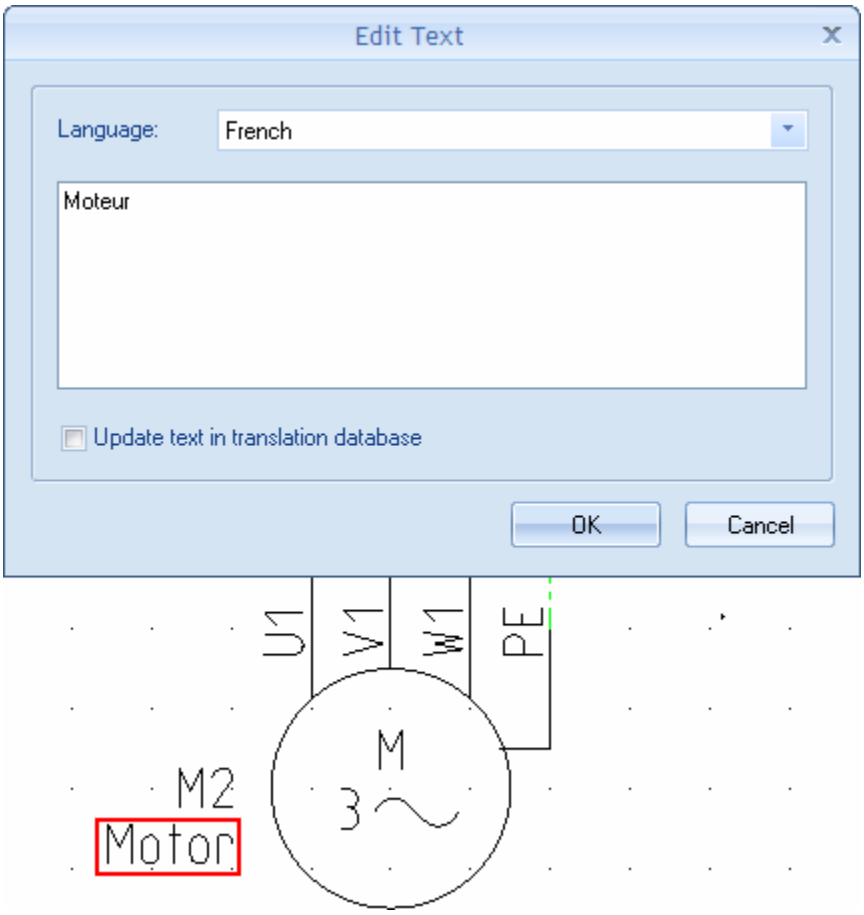
Diagram: Source text	*French translation*
Motor 1	*Moteur 1*
Motor 2	*Moteur 2*
Motor control	*Controle du moteur*
Motor control circuit	*Controle du moteur circuit*

EE.8.3. UPDATING TRANSLATION TEXT IN A SHEET

(Advanced)

Generally, when you modify an already translated text, you can only change the text in the source language. In the example below the text "Motor" has been translated from English to French – Moteur.
The **Functions** □ **Other** □ **Update Translated Text** command allows you to directly edit the translation text.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



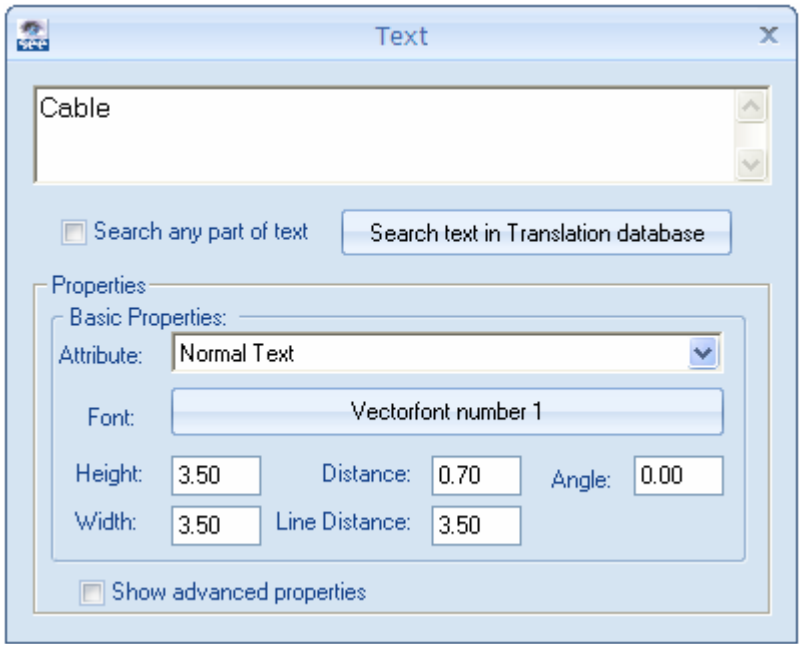
Exercise 29-2: Update a translated text in your workspace.

1.CO **Functions □ Other □ Update Translated Text**
2. Select the text whose translation you wish to update.
3. Select the language from the "***Language***" pull-down list in the dialogue that appears.
4. Fill in the new text.
   An additional option allows you to modify or add a text into the translation database.
5.> **OK**
   The modified translation is inserted into the translation database.

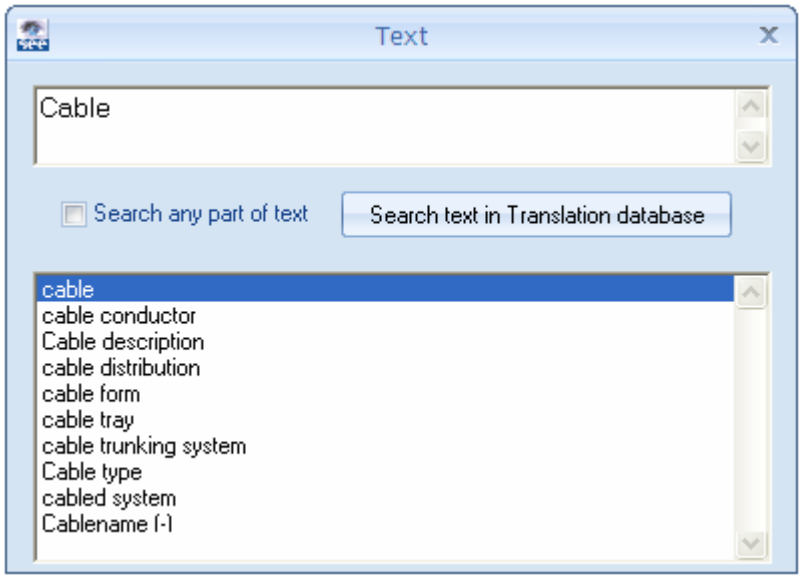
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

EE.8.4. LOOKING UP TEXTS IN THE TRANSLATION DATABASE

If you use a text frequently, it must be written at all the places in the same way.
When you use the **Draw** □ **Elements** □ **New** or **Edit** □ **Text** □ **Edit Text** commands, you can look up in the translation database while entering the text if the text is available and how it was written exactly. In this way you can avoid different spellings and reduce errors.



Type the text and click the **Search text in Translation database** button. All the texts from the translation database ("***Source language***" column) beginning with this text are displayed and can be selected by double-clicking with the mouse.
If you activate **Search any part of text**, the whole phrase is searched in the translation database. For the text "Cable", the "cable channel" term will also be found.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

EE.8.5. EXCHANGING THE SOURCE LANGUAGES OF A WORKSPACE

(Advanced)

Through the **TransText2SourceText** and **TransText2SourceTextWsp** commands, available in the ***Commands*** pane you can transfer one existing translation language as a source language in a single page or a workspace.

The **SwapSourceLanguageInTranslationDatabase** command gives you the possibility to define as a source language each language available in the current translation database (*TranslationNew.MDB* in the *Templates* folder of your *SEE Electrical* installation).

If the new source language contains empty or duplicated entries, there will be a loss of data, because the entry in the source language column cannot be empty; if one term is found twice in the new source language, the second entry will overwrite the first one or it will be refused.

Make sure to avoid both situations before you activate the command.

Within the translation database you can manage twenty one languages including the source language.

Exercise 29-1: Swap the languages in the translation database.

Make a backup copy of your existing translation database (*TranslationNew.MDB* in the *Templates* folder of your *SEE Electrical* installation).

1.CO **SwapSourceLanguageInTranslationDatabase**
2. Select the language that shall be the new source language in the "***Select language to be Source Language***" field.
3. > Type the language name of the new source language.
Make sure to use a name not already used as a language column. If you have to use an already existing name, it will be used only in case it does not contain entries. If it contains entries, an error message will appear and you will have to delete the entries before you could use the language.
If you use the name of an existing column, a new column ("***Language New***") will be generated in order to keep the amount of twenty one languages in the language database. In case you have duplicated or empty entries in the new source language, a list of these entries appears. A *SwapTranslationSourceLanguage.log* file is created in the *..\Templates* folder that contains the list of entries.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

EE.9. CHANGING THE FONTS AND ATTRIBUTES FOR ALL TEXTS IN THE WORKSPACE, SYMBOL LIBRARIES OR PAGE TEMPLATES

You can change the fonts in the workspaces, page templates and symbol libraries. All tools are located in the ***Commands*** pane of the software. It is possible to change the fonts for a single project or for several project, symbol libraries and templates.

Exercise 29-3: Finding and changing all fonts used in a single workspace.

1.CO **FontToolReadSingleProject**
2. Define the project to scan for fonts - the current project is suggested by default.
3. Press the ... button to choose another project.
4.> Define the XML file to write the font settings into in the "***Set XML file to export text***" field.
5. Press the ... button to open and load the desired folder and then type name of *XML* file.
6. Close the folder window with the help of the **Open** button.
   If you select an existing *XML* file, a question appears asking you if you want to add the results of the new scan to this *XML* file or if you want to start again. If you add the new results to the existing file, you can benefit from the mappings already done and you will be sure that you use the same definitions for all your data.
7. Press the **Scan project for texts** button to start the scanning process.
8. CO **FontToolMapFont**
9. Choose the first *XML* file with the results from the scanning of your data.
10. Change the settings for "***New Font Name***", "***New Font Height***", "***New Font Width***" and "***New Font Distance***" for each of the different fonts and the combinations of text attributes.
    In case you need to use a different code page than the default one, it is possible to choose the appropriate script when defining the font. The script you defined appears in the "*New Font Script*" field but cannot be changed
11. Save your changes with the help of the **Set font mapping** button.
12. Close the window with **Cancel.**
13. CO **FontToolChangeSingleProject**
14. Define the project to replace the fonts in - the current project is suggested by default.
15. Press the ... button to choose another project.
16. Choose the XML file to take the mapping information from in the "***Set XML file to export text***" field.
17. Press the ... button to open and load the desired folder and then select the *XML* file.
18. Close the folder window with the help of the **Open** button.
19. Press the **Replace projects text** button to start the replacing process.

Exercise 29-4: Finding and changing all fonts used in several workspaces.

1.CO **FontToolReadProjects**
2. Press the button within the ***Projects*** area to define the projects to scan for fonts.
3. Press the button if you want to remove a project from the list.
4.> Define the *XML* file to write the font settings into in the "***Set XML file to export text***" field.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

5. Press the button to open and load the desired folder and then type name of XML file.
6. Close the folder window with the help of the **Open** button.
   If you select an existing *XML* file, a question appears asking you if you want to add the results of the new scan to this *XML* file or if you want to start again. If you add the new results to the existing file, you can benefit from the mappings already done and you will be sure that you use the same definitions for all your data.
7. Press the **Scan projects for texts** button to start the scanning process.
8. CO **FontToolMapFont**
9. Choose the first *XML* file with the results from the scanning of your data.
10. Change the settings for "***New Font Name***", "***New Font Height***", "***New Font Width***" and "***New Font Distance***" for each of the different fonts and the combinations of text attributes.
    In case you need to use a different code page than the default one, it is possible to choose the appropriate script when defining the font. The script you defined appears in the "*New Font Script*" field but cannot be changed
11. Save your changes with the help of the **Set font mapping** button.
12. Close the window with **Cancel.**
13. CO **FontToolChangeProjects**
14. Press the button within the ***Projects*** area to define the projects to scan for fonts.
15. Press the button if you want to remove a project from the list.
16. Choose the *XML* file to take the mapping information from in the "***Set XML file to export text***" field.
17. Press the button to open and load the desired folder and then select the *XML* file.
18. Close the folder window with the help of the **Open** button.
19. Press the **Replace projects text** button to start the replacing process.

Exercise 29-5: Finding and changing all fonts used in a single or in several libraries.

1.CO **FontToolReadSymbol**
2. Press the button within the ***Symbols*** area to define the libraries to scan for fonts.
3. Press the button if you want to remove a library from the list.
4.> Define the *XML* file to write the font settings into in the "***Set XML file to export text***" field.
5. Press the button to open and load the desired folder and then type name of *XML* file.
6. Close the folder window with the help of the **Open** button.
   If you select an existing *XML* file, a question appears asking you if you want to add the results of the new scan to this *XML* file or if you want to start again. If you add the new results to the existing file, you can benefit from the mappings already done and you will be sure that you use the same definitions for all your data.
7. Press the **Scan symbols text** button to start the scanning process.
8. CO **FontToolMapFont**
9. Choose the first *XML* file with the results from the scanning of your data.
10. Change the settings for "***New Font Name***", "***New Font Height***", "***New Font Width***" and "***New Font Distance***" for each of the different fonts and the combinations of text attributes.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

In case you need to use a different code page than the default one, it is possible to choose the appropriate script when defining the font. The script you defined appears in the "*New Font Script*" field but cannot be changed

11. Save your changes with the help of the **Set font mapping** button.
12. Close the window with **Cancel**.
13. CO **FontToolChangeSymbol**
14. Press the button within the ***Library*** area to define the projects to scan for fonts.
15. Press the button if you want to remove a library from the list.
16. Choose the *XML* file to take the mapping information from in the "***Set XML file to export text***" field.
17. Press the button to open and load the desired folder and then select the *XML* file.
18. Close the folder window with the help of the **Open** button.
19. Press the **Replace symbols text** button to start the replacing process.

Exercise 29-6: Finding and changing all fonts used in a single or in several templates.

1.CO **FontToolReadTemplate**
2. Press the button within the ***Templates*** area to define the templates to scan for fonts.
3. Press the button if you want to remove a template from the list.
4.> Define the XML file to write the font settings into in the "***Set XML file to export text***" field.
5. Press the button to open and load the desired folder and then type name of *XML* file.
6. Close the folder window with the help of the **Open** button.
   If you select an existing *XML* file, a question appears asking you if you want to add the results of the new scan to this *XML* file or if you want to start again. If you add the new results to the existing file, you can benefit from the mappings already done and you will be sure that you use the same definitions for all your data.
7. Press the **Scan templates text** button to start the scanning process
8. CO **FontToolMapFont**
9. Choose the first *XML* file with the results from the scanning of your data.
10. Change the settings for "***New Font Name***", "***New Font Height***", "***New Font Width***" and "***New Font Distance***" for each of the different fonts and the combinations of text attributes.
    In case you need to use a different code page than the default one, it is possible to choose the appropriate script when defining the font. The script you defined appears in the "*New Font Script*" field but cannot be changed
11. Save your changes with the help of the **Set font mapping** button.
12. Close the window with **Cancel**.
13. CO **FontToolChangeTemplate**
14. Press the button within the ***Templates*** area to define the templates to scan for fonts.
15. Press the button if you want to remove a template from the list.
16. Choose the *XML* file to take the mapping information from in the "***Set XML file to export text***" field.
17. Press the button to open and load the desired folder and then select the *XML* file.
18. Close the folder window with the help of the **Open** button.
19. Press the **Replace templates text** button to start the replacing process.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

EE.10. PLC FUNCTIONALITIES IN THE ADVANCED LEVEL

Automatic numbering of addresses

If you use *SEE Electrical advanced*, you can set the PLC Address Numbering Method in the ***Circuit Diagrams Properties*** (Decimal, Octal, Hexadecimal (i.e. lower case letters in addresses, for example E0.a) or HEXADECIMAL (i.e. upper case letters in addresses, for example E0.A)). If you enter a value for an address (for example, in the Rack), the next values based upon it are automatically numbered.

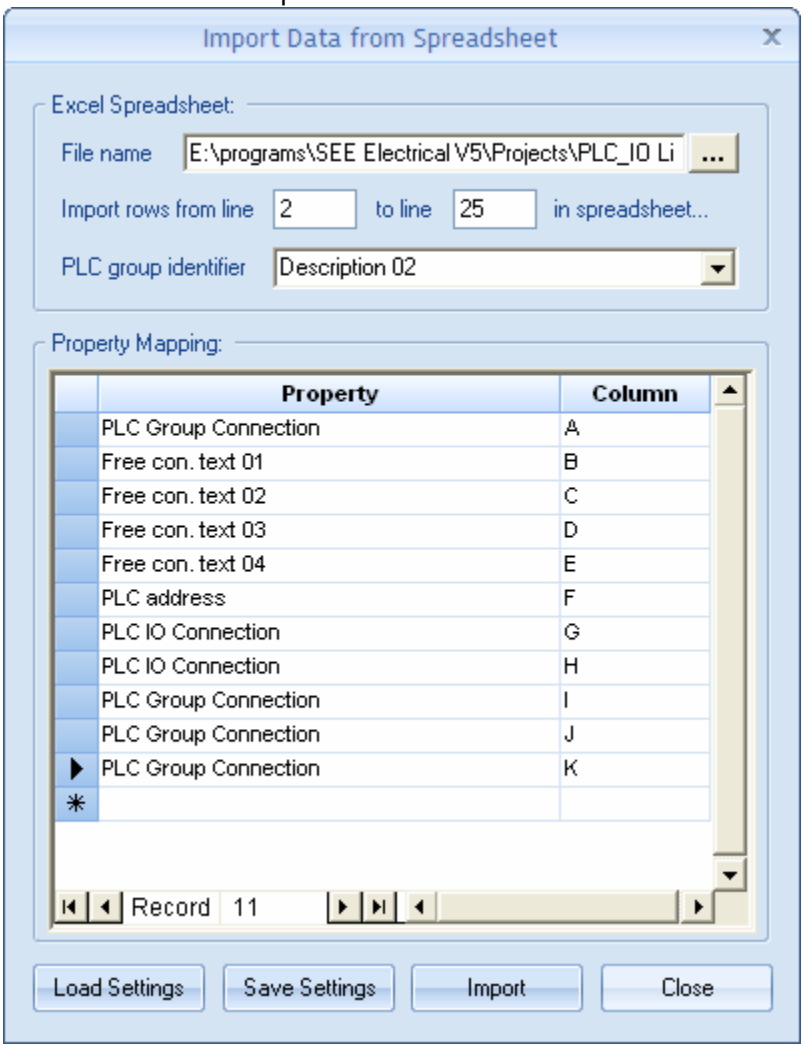
So if you need to position a lot of addresses on the page, position the first one, give the good name of the component and the address. Then copy the next ones. The addresses are automatically numbered according to the chosen PLC Address Numbering Method.

Import the PLC allocation list

There is a feature in *SEE Electrical* for importing a PLC allocation list if it is available as an Excel file. PLC allocation lists usually do not contain information about the component name of the PLC in the Circuit diagram. Instead of this, the PLC components are identified via module names. For this reason, the module names are taken into consideration while importing the allocation list.

Import the Excel List

The import of the Excel list in the workspace can be done by executing the PLCImportExcel function in the ***Commands*** explorer.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Define the text property for the group identifier of the PLC. This text property must be available in the list of the importable text properties.

Symbols

The PLC-Rack symbols and PLC-connections symbols must contain all the texts that are imported, i.e. PLC-Rack symbols and PLC-connections symbols must contain in particular the "PLC group identifier" and the "PLC-address" connections. For the import of the allocation list, it must be chosen which text should receive the "PLC group identifier" (for example, Free text 01 or Free text 20 or Description). It is important to decide which text is possible and you have to insert this text in your PLC Rack and PLC connections.

Assign the information to the symbols

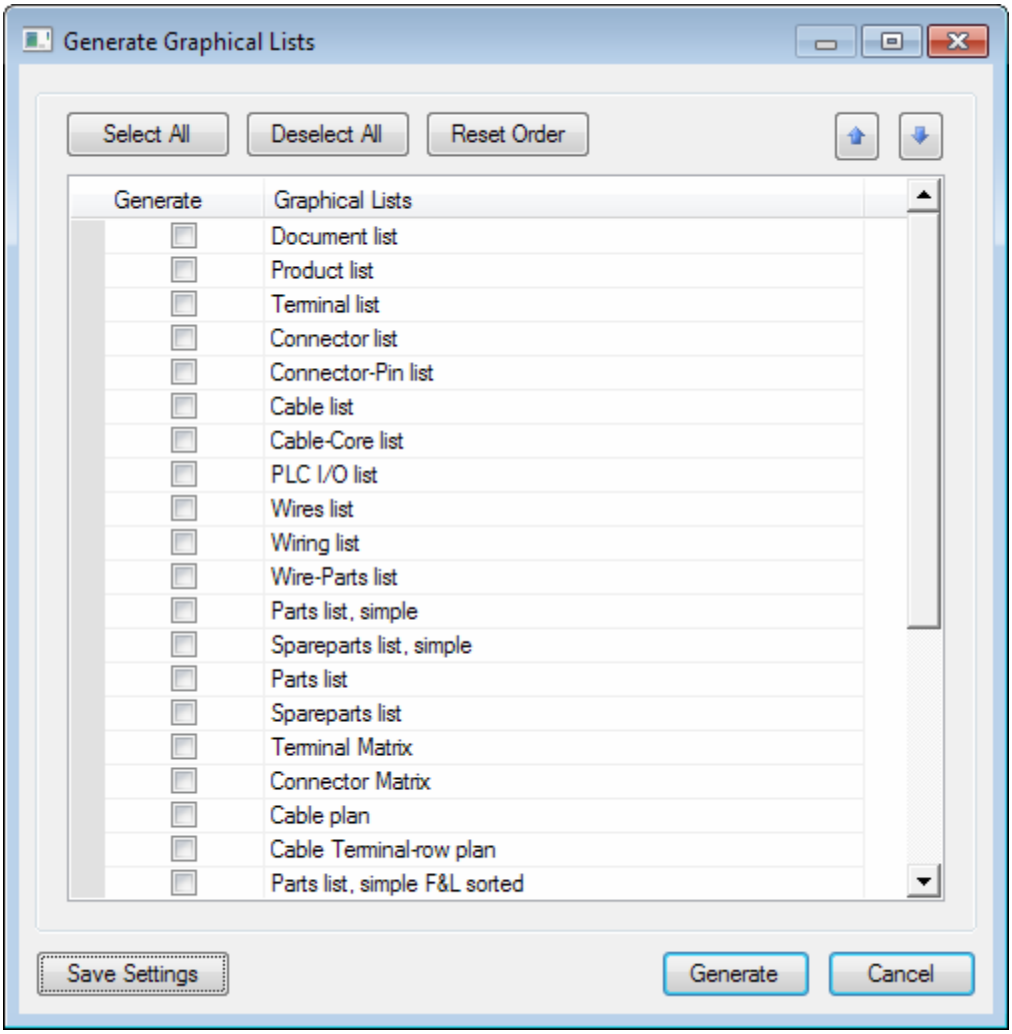
Double-click the **PLCImportAssign** command from the ***Commands*** explorer to execute it. Afterwards identify the symbol that you want to assign information to.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

EE.11. GENERATE GRAPHICAL LISTS IN ONE STEP

Advanced

The *Graphical lists* can be generated in one step using the **Generate** command in the *Graphical lists* node.



Exercise 29-7: Generate the graphical lists with the help of the **Generate** pop-up command:

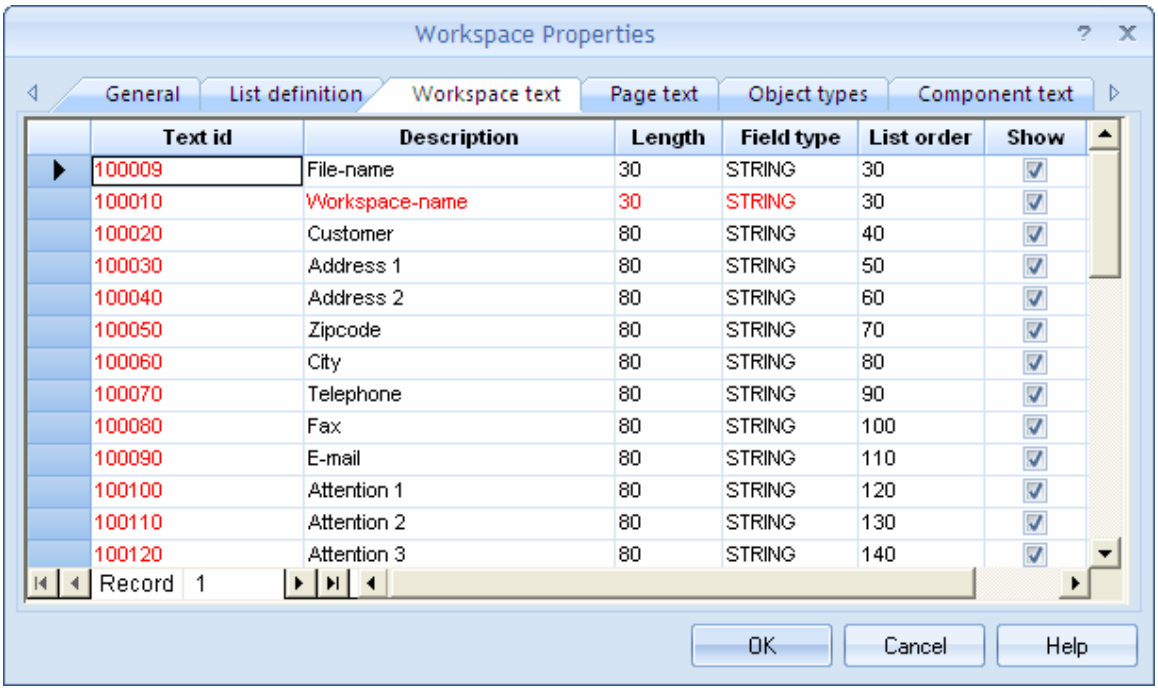
1.+ In the ***Workspace Explorer***, right-click the *Graphical lists* node.
2.CO **Generate**
3.> Select the lists you want to generate.
4.> The button allows you to define the order in which the lists are generated. You can make sure that the document list is generated as last one so it will contain all others.
5.> You can use the **Save Settings** button to save the settings you just defined for using them again.
6.> **Generate**

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

FF CUSTOMIZING THE WORKSPACE/PAGE INFORMATION WINDOWS

You can change the texts in the Workspace information and Page information dialogue boxes. For example, Project-description line 3 could become Commission number. You can change the sequence of displaying the texts in the windows, hide records, or add your own records.

Right-click the workspace name in the ***Workspace Explorer*** and select the **Properties** pop-up command. You can find the options for changing texts in the ***Workspace text*** tab or the ***Page text*** tab.



Records highlighted in red can be changed.

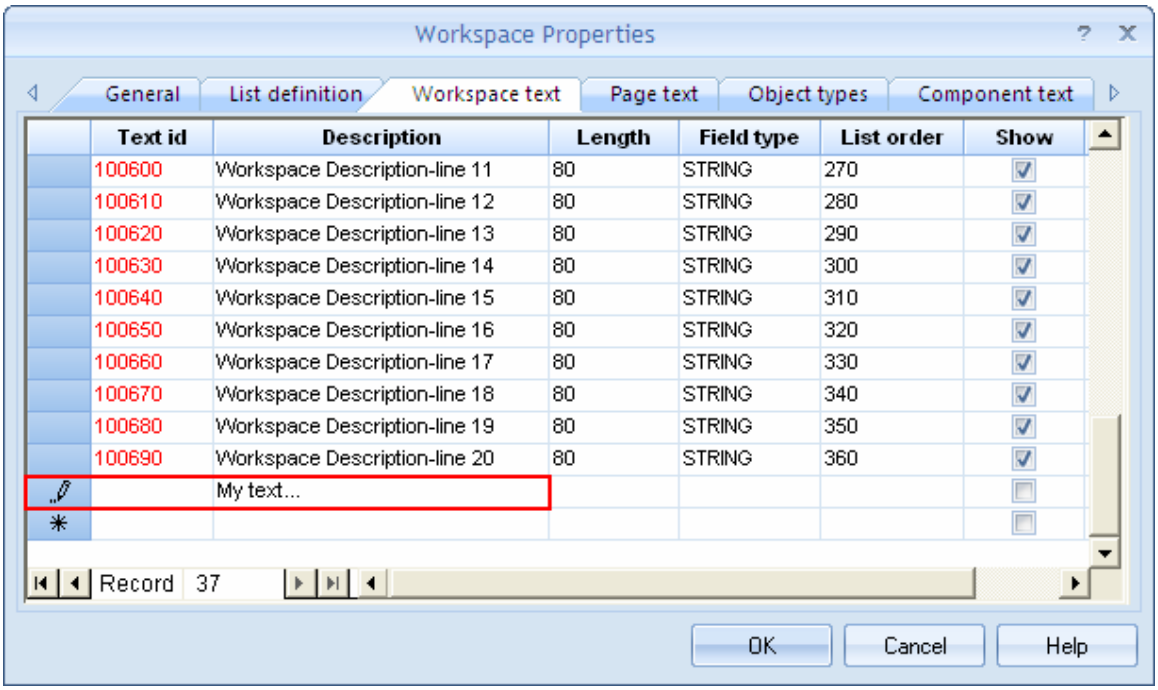
The IDs for user-defined Workspace texts must be within the range 102000 to 110000, and the page texts –within the range 122000 to 130000.
The example below illustrates how a user-defined workspace text is created. Similar procedures are applicable for the creation of page texts and component texts.

Example:

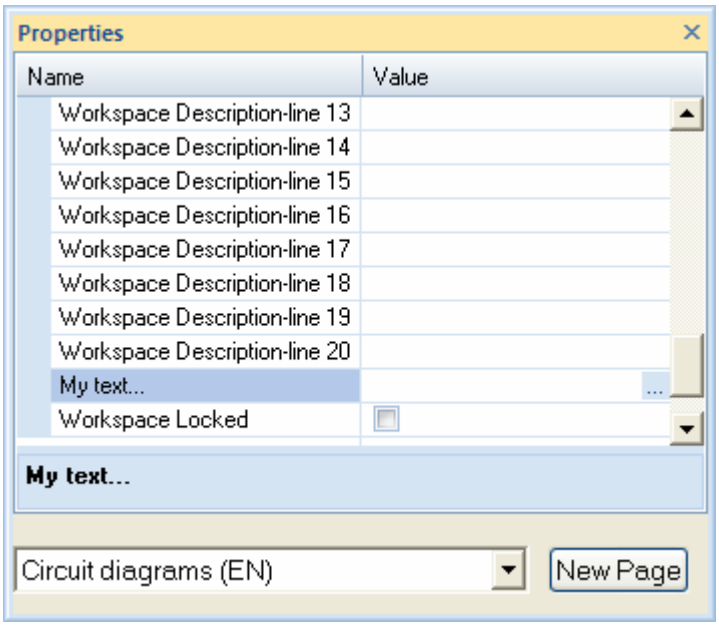
*1. Open the **Workspace Properties** window in the already described way and click the **Workspace text** tab.*
2. Scroll down to the very bottom of the window and click in the empty Description field.
3. Type in your text there, for example: "My text…".

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

A new empty line is automatically inserted:



*4. Fill in the other fields as desired, tick the check box in the "Show" column and click **OK** to apply your new text.*
A message appears informing you that you have to close the workspace and open it again in order to activate the new setting(s).
5. Close the workspace and re-open it.
*The new text is displayed in the corresponding **Workspace information** pane:*



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Hints

*1. If you click **OK** without entering the **Text id** field, a message appears, informing you that your text ID must be within a certain range.*
*2. To delete any user-defined entry, select it and press the **Delete** key on the keyboard.*

You can also hide not needed Database lists and Graphical lists.
You can find the customizing options for the workspace tree within the ***Workspace Properties*** dialogue box by clicking the ***List definition*** tab.

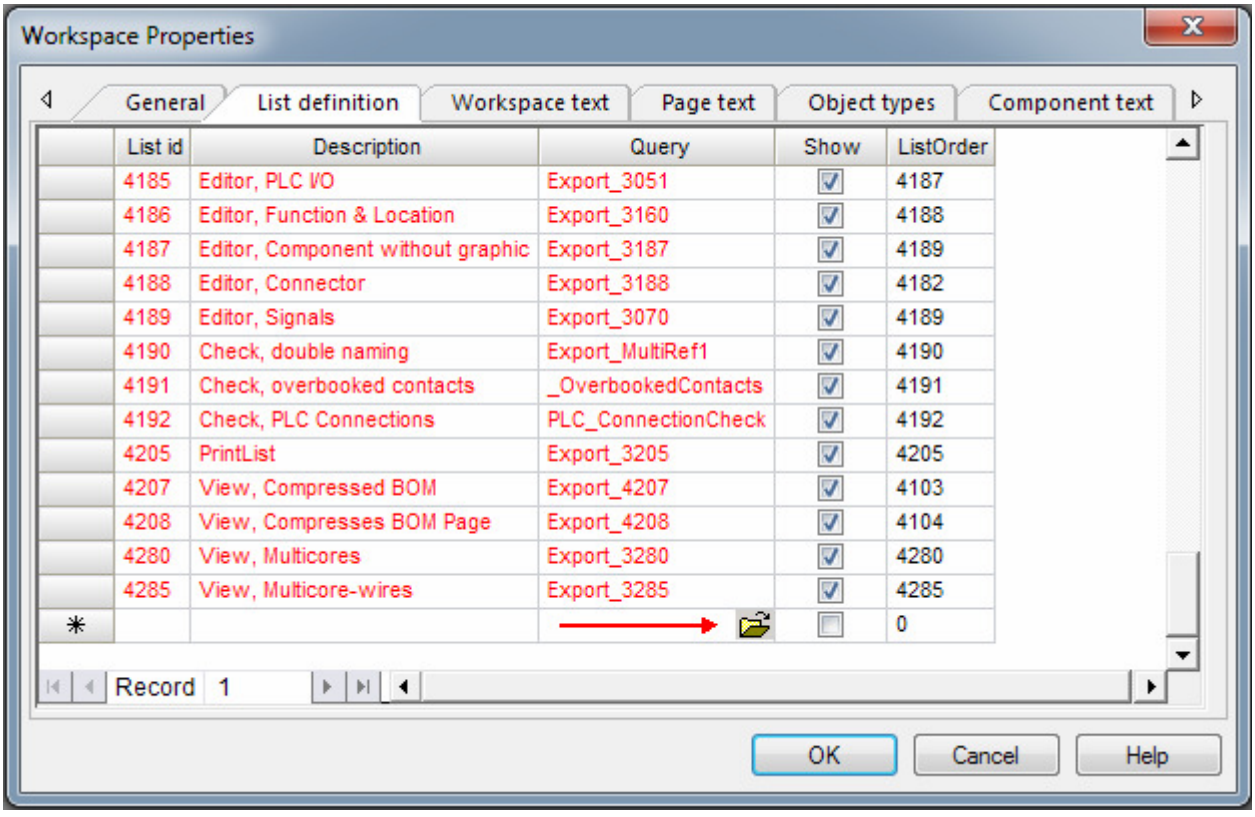
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

GG CUSTOMIZING THE WORKSPACE TREE

(Advanced)

GG.1. HIDE UNNECESSARY DATABASE AND GRAPHICAL LISTS

It is possible to create your own database and graphical lists, and thus your own SQL queries. (However, it is not possible to insert the "Terminal Matrix", "Terminal Row Picture", "Terminal Row Picture Plan" or "Cable Plan" graphical lists for a second time into the project tree. Also database editors are not user-defined customizable.)
In the ***List definition*** tab of the ***Workspace Properties*** window, you can find the options for customizing.

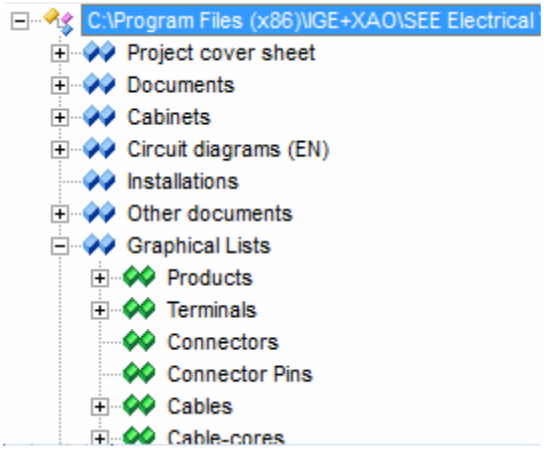


User defined database lists must contain a number from the range 4501 to 4999. The numbers for custom graphical lists must be between 3501 and 3999.

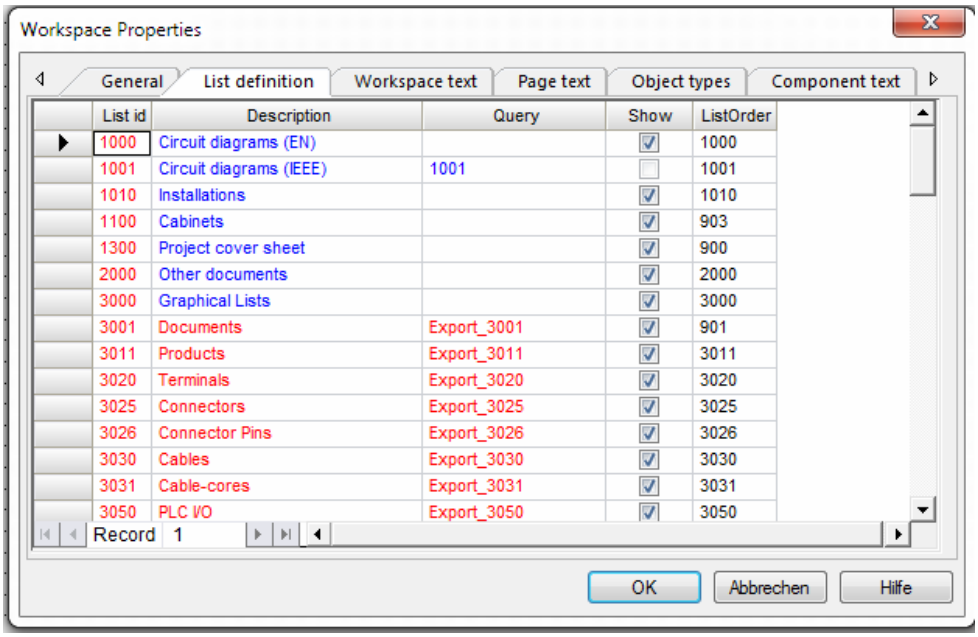
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

GG.2. SORTING OF DOCUMENTS IN THE WORKSPACE TREE

You can change the order of appearance of the lists in the workspace tree. For example, you can order the workspace tree in the following way:



To sort the lists as desired, change the list order number in the respective field:



For example give 1 for the Project cover sheet, 2 for the Document list, 3 for the cabinets and 4 for the Circuit Diagrams.

The native sort orders for the grafical lists are values between 3000 and 3999.

The database lists can be sorted using values between 4000 and 4999.

The sort order given here is the one used in the **Print** command when all pages are printed one after the other.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

The "*View, Documents*" and "*Editor, Documents*" database lists show the documents as ordered in the workspace tree.

The order of the documents part of the *Graphical lists* depends on the sorting order within the ***List Definition*** tab, e.g if the order is the default one (3001) or a customized order is used. If the default order has not been changed, for compatibility reasons, the *Documents list* is always inserted in the top of the lists and you can still use the **FirstContentLine** command.

If you use a customized order, the *Documents list* is sorted according to the workspace sort order. If you want to keep the *Documents list* in first place in the *Graphical lists* and it has to be in first place also in the *Documents* node, you could change the sort order to 3002.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

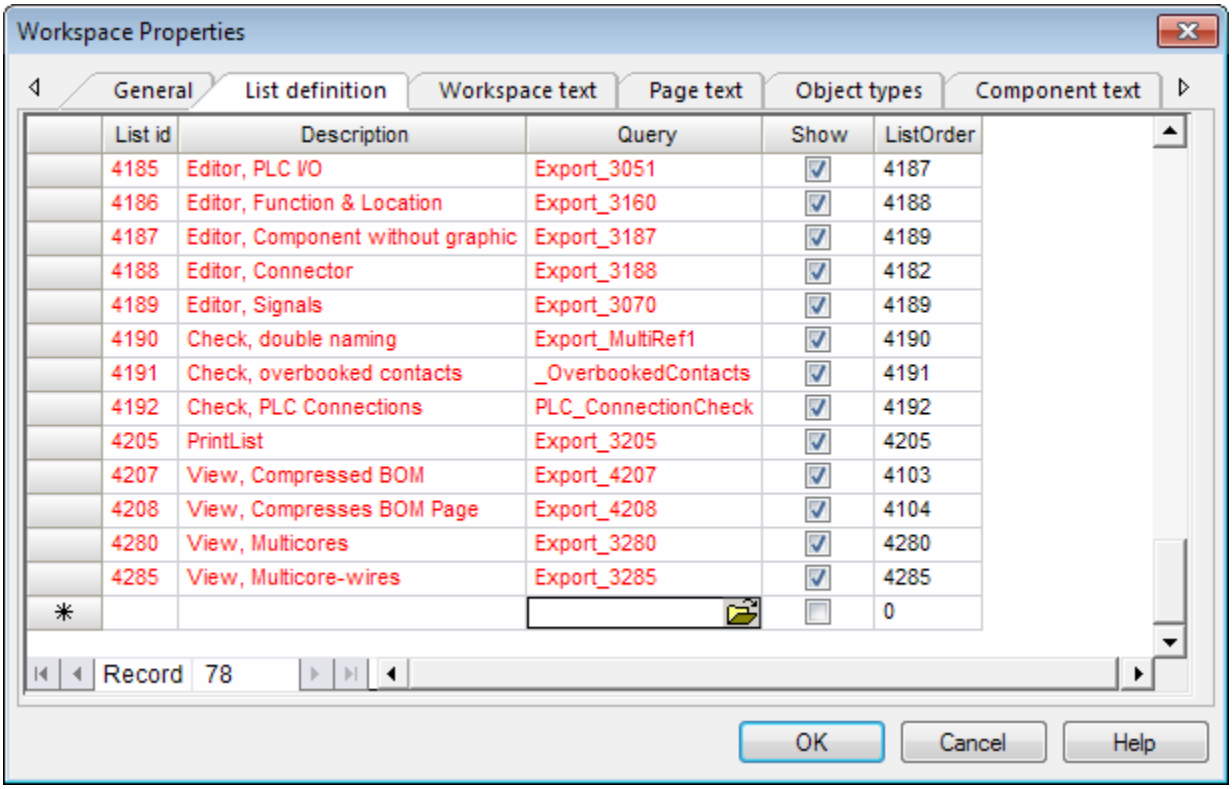
HH USER-DEFINED SQL-QUERIES

(Advanced)

SQL is embedded in the *SEE Electrical* database to allow you to create any kind of form, including summarization of different parameters.
You have the possibility to create your own Database lists and Graphical lists, and herewith your own *SQL*-queries. (However, it is not possible to add the "Terminal matrix", "Terminal plan", "Terminal plan (multi level)" or "Cable plan" into the Workspace Explorer again. Database editors are not customizable either.)
SQL-queries are created or modified by means of the *SQL* Builder.

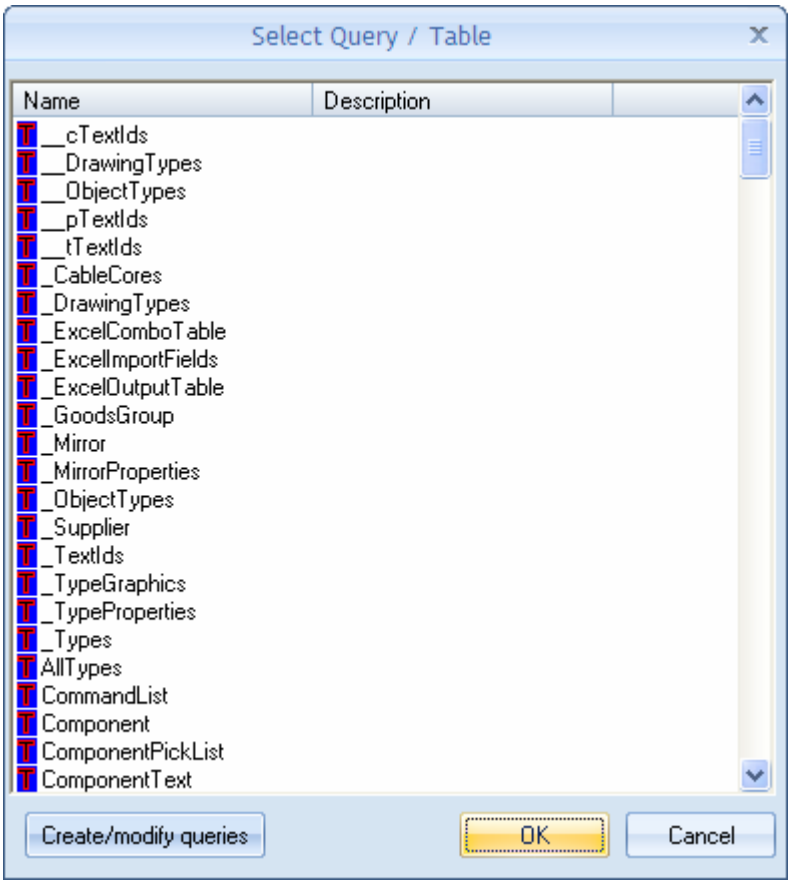
HH.1.1. CREATING AN SQL-QUERY FOR ADDING A DATABASE LIST IN THE WORKSPACE EXPLORER

For accessing the *SQL* Builder, right-click first on the workspace name in the ***Workspace Explorer*** and select the **Properties** pop-up command to open the ***Workspace Properties*** window. Then click the icon in an empty Query field within the ***List definition*** tab:



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

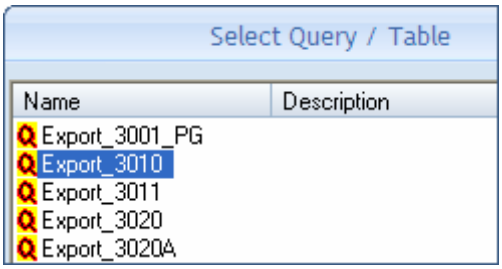
The following dialogue appears:



When you click the **Create/modify queries** button, the *SQL* Builder opens. Here, after clicking the **Select table** / **query** button, the dialogue you can see above opens again. Now you can define your own query.

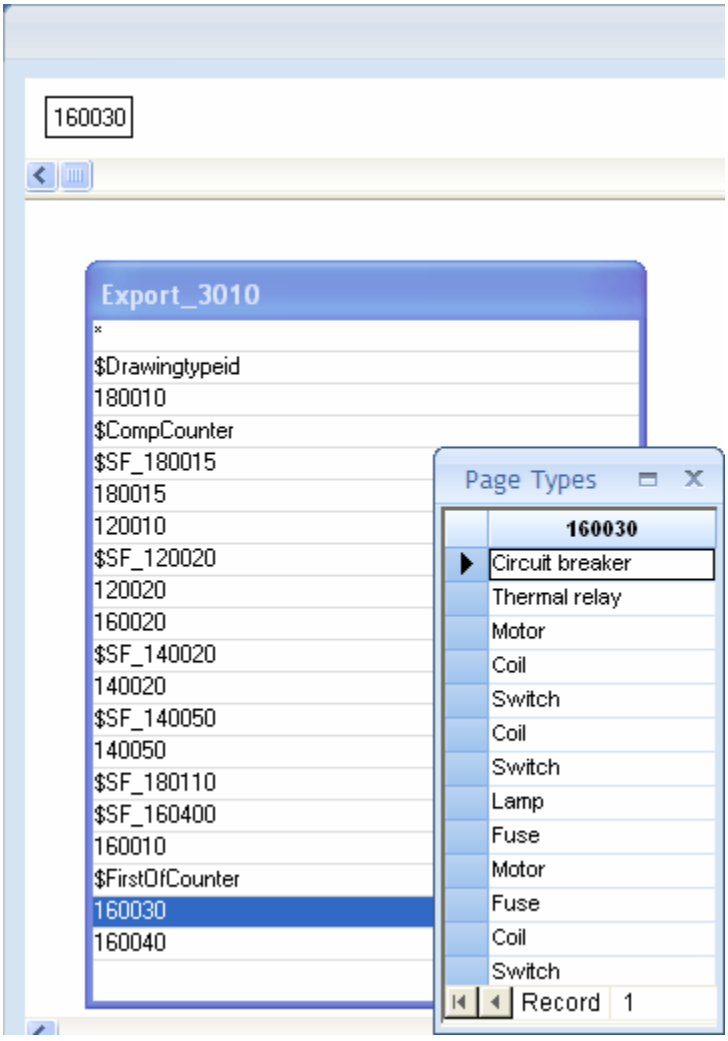
Example of creating a list with a total amount of prices:

*Select, within the **Select Query** / **Table** window, the Export_3010 query, for example, and click **OK**:*



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

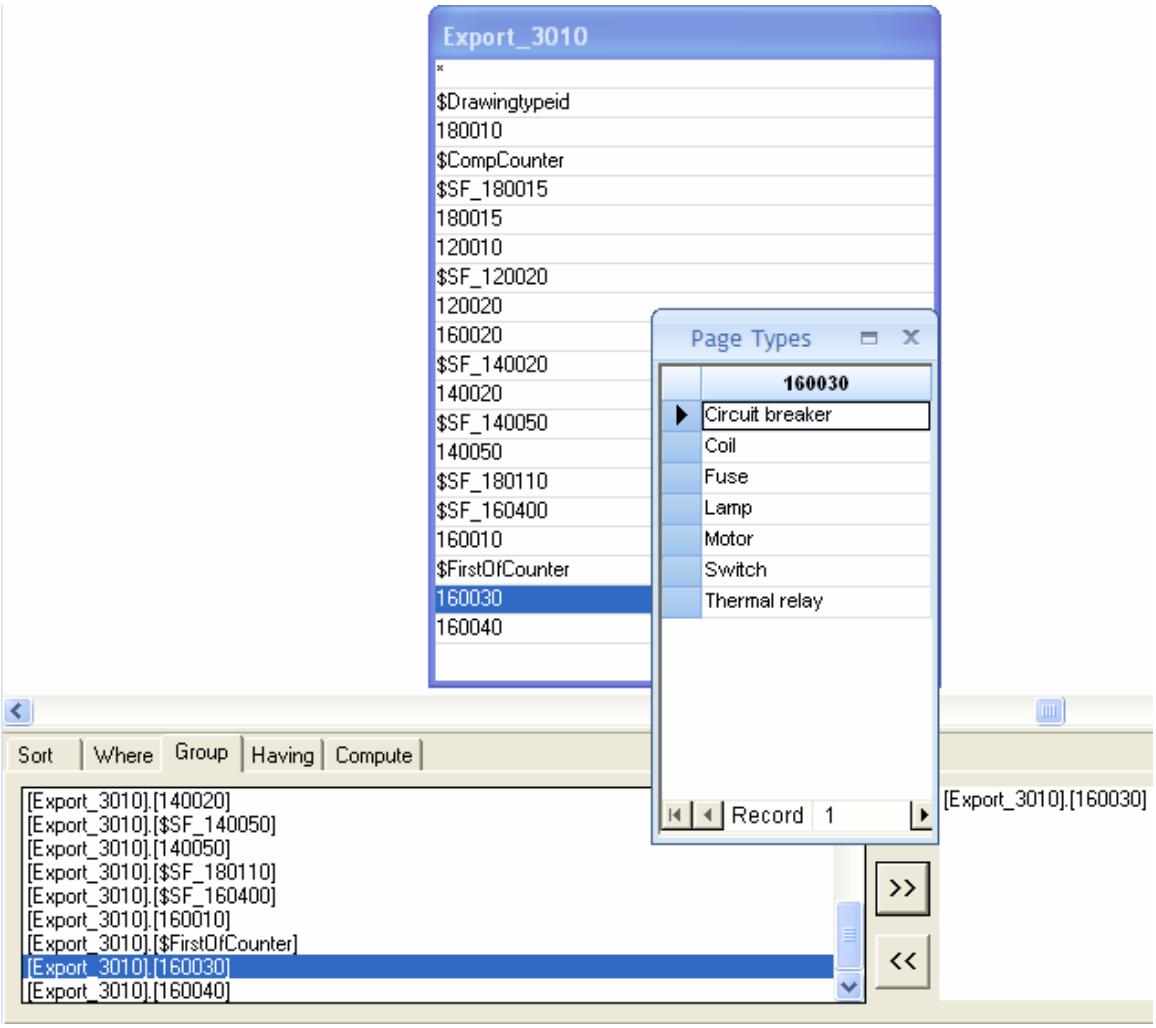
*From the Export_3010 table that appears, select 160030 which is the ID of Description 00, and click the **Preview** / **Test Query** button in the SQL Builder. A table is opened that contains the "Description 00" column from the Products database list with all entries from your project:*



*Close the **Page Types** dialogue.*
Now, group the entries first, i.e. create a column containing all entries only once.
*For this purpose, within the **Group** tab, select [Export_3010].[160030] and move it to the right bottom pane using the corresponding directional button.*

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

*Then click the **Preview** / **Test query** button. The **Page Types** dialogue appears again but now containing all the entries grouped – no multiple entries are displayed:*



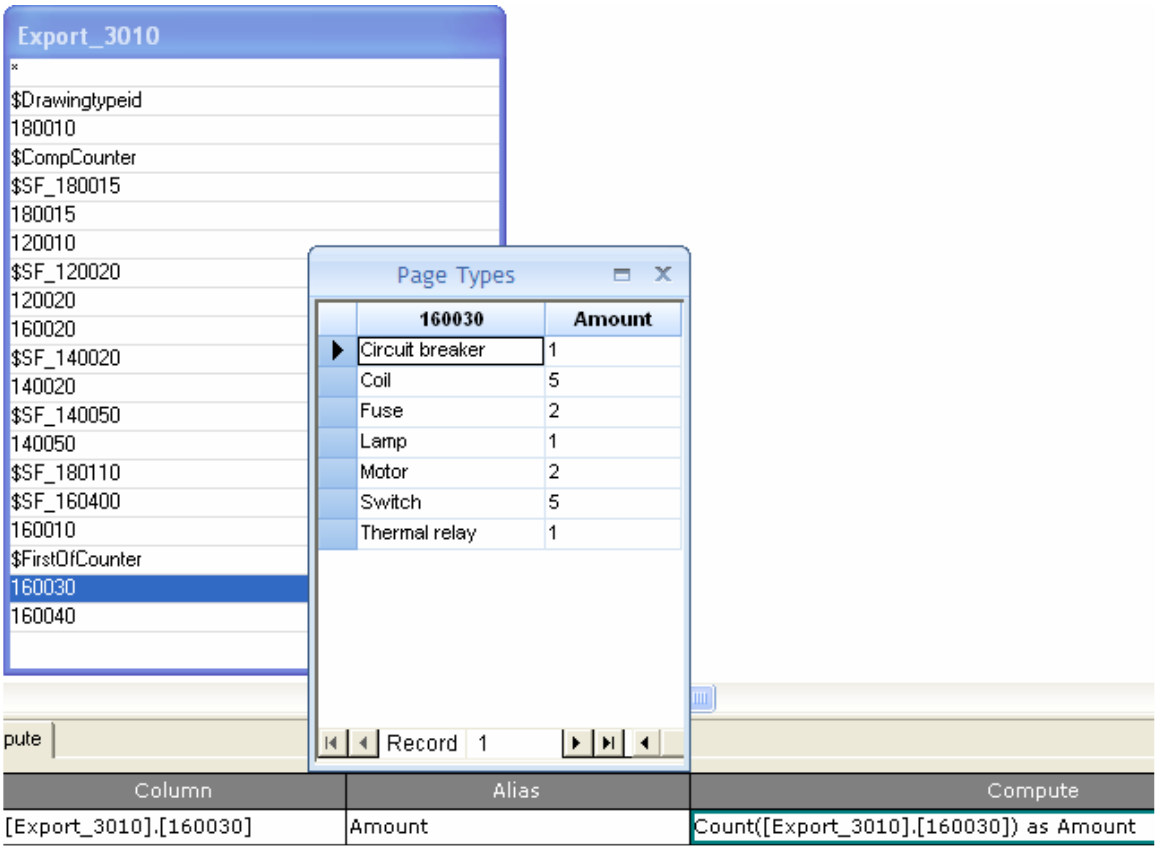
The next step is to calculate the total number of each entry (item) which will be displayed in another column next to the respective item entry to show what the existing quantity of that is.
*So, close the **Page Types** dialogue and click the **Compute** tab.*

*Within the" **Function**" field, select Count from the scroll-down list that opens.*
*Then click within the "**Column**" field and select [Export_3010].[160030] from the pull-down list.*
*In the **Alias** field, type in, ex. Amount for the type.*

*Select afterwards the "**Compute**" field being already filled in, and then click the **Preview** / **Test Query** button.*

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

*The **Page Types** dialogue again appears but now showing also the number of each kind of item, i.e. the amount of each grouped entry. As well, this is the way in which the respective database list will look within SEE Electrical:*



*Close the **Page Types** dialogue.*

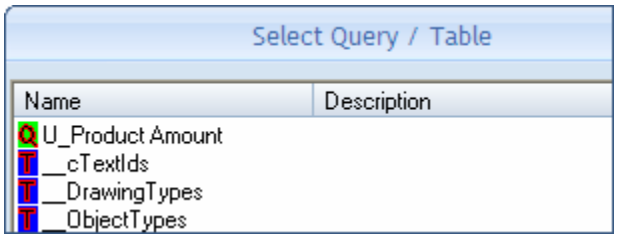
Hint

---

*If you want to view the string of your query, click the **Show SQL string** button: a small message dialogue appears displaying your SQL-query string. Click **OK** to close it.*

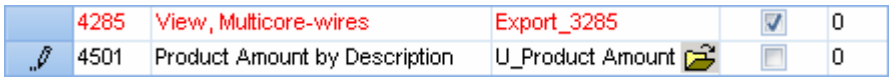
- Click the **Save query** button to save your new defined *SQL*-query, give it a relevant name, ex. **Product Amount**, and exit the *SQL* Builder.

Your query is added to the list in the ***Select Query / Table*** dialogue window:



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

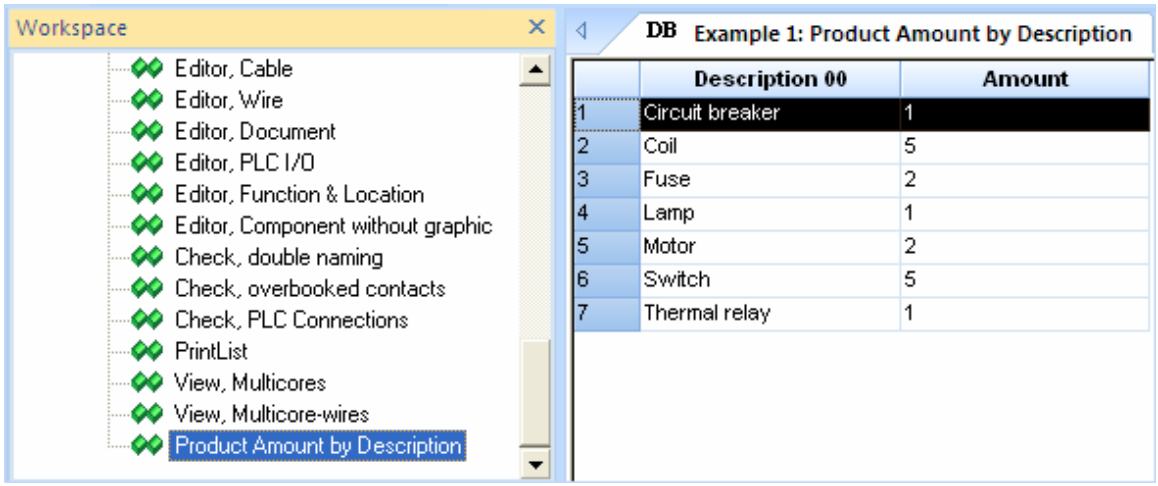
- Define now a new **List Definition** (Database list) in the ***Workspace Properties*** window and choose for query the one just created.



- Tick the check box in the "***Show***" column and click **OK** to save the changes.
- Close the workspace. When you re-open it, the new database list will be displayed in the ***Workspace Explorer***.

User-defined Database Lists must receive an ID number within the range 4501 –4999. The number for user-defined Graphical lists must be within the range from 3501 to 3999.

In this new database list created with the user-defined query described above, all Descriptions 00 concerning the various items are listed and what is more, all products with the same description are counted and their total number is displayed for each description:



HH.1.1. DEFINITION OF GRAPHICAL FORMULAS

Within the graphical lists, the query is inserted as a normal text containing the following syntax: #QUERY [<name of query>] #WHERE [<argument>]="<content>".

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

For example: #QUERY [U_APd] #WHERE [APno]="1"

B: Power electrical equipment

Description of equipment			Power (KW)
#QUERY [U_APd]	#WHERE [APno]="1"	#QUERY [U_APp]	#WHERE [APno]="1"
#QUERY [U_APd]	#WHERE [APno]="2"	#QUERY [U_APp]	#WHERE [APno]="2"
#QUERY [U_APd]	#WHERE [APno]="3"	#QUERY [U_APp]	#WHERE [APno]="3"
#QUERY [U_APd]	#WHERE [APno]="4"	#QUERY [U_APp]	#WHERE [APno]="4"
#QUERY [U_APd]	#WHERE [APno]="5"	#QUERY [U_APp]	#WHERE [APno]="5"
#QUERY [U_APd]	#WHERE [APno]="6"	#QUERY [U_APp]	#WHERE [APno]="6"
#QUERY [U_APd]	#WHERE [APno]="7"	#QUERY [U_APp]	#WHERE [APno]="7"
#QUERY [U_APd]	#WHERE [APno]="8"	#QUERY [U_APp]	#WHERE [APno]="8"
#QUERY [U_APd]	#WHERE [APno]="9"	#QUERY [U_APp]	#WHERE [APno]="9"
#QUERY [U_APd]	#WHERE [APno]="10"	#QUERY [U_APp]	#WHERE [APno]="10"

Summary Power: electrical equipment #QUERY [U__APtotPW]

Maximum 10 pieces of equipment can be listed in the list in this example. The index for the components listed in this list has to be unique, as we want to see each piece of electrical equipment in one line.
This template generates the following result:

B: Power electrical equipment

Description	Power (KW)
cooker	7.50
Boiler	4.00
washing machine	2.50
air conditioner	1.50

Summary Power: electrical equipment 15.50 KW

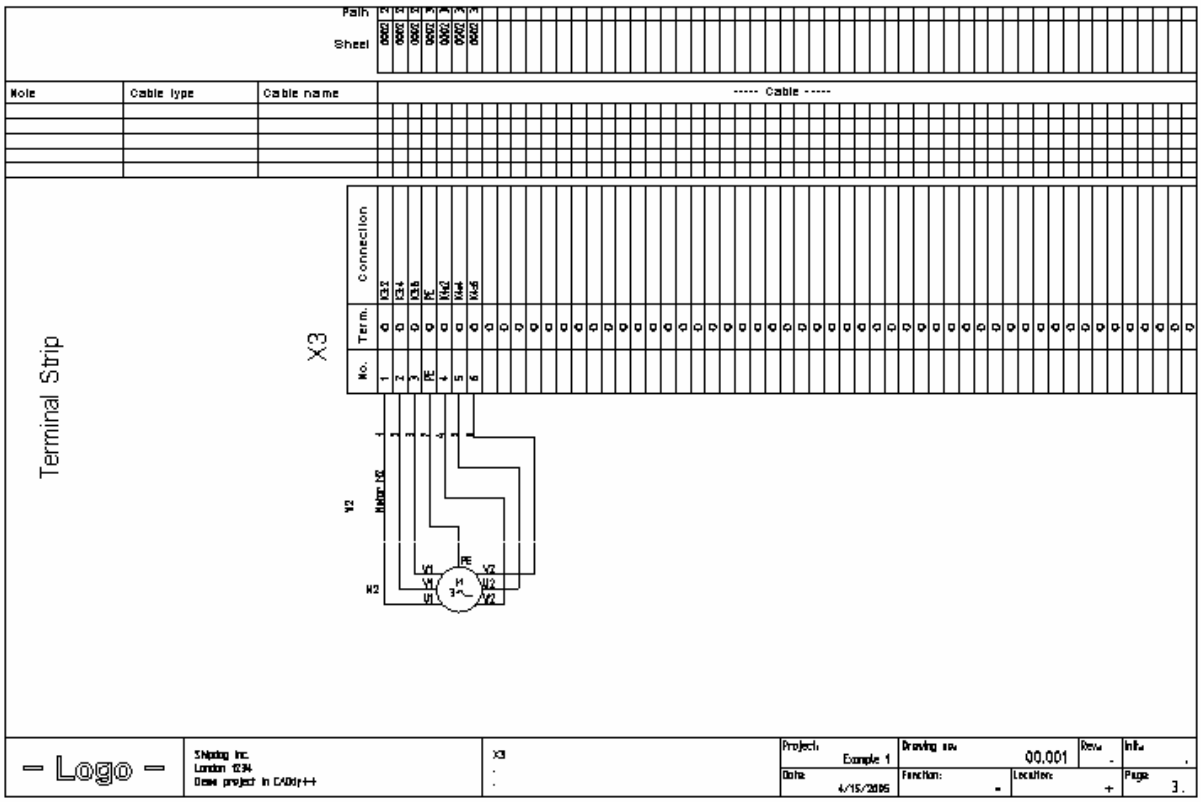
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

II TERMINAL PLAN

(Advanced)

II.1. GENERATING A TERMINAL PLAN

In terminal plans with graphics, the first target is assigned to each terminal as a symbol; the second target is assigned to the terminal as a text.



Exercise 32-1: Generate a terminal plan for the training project.

1. Select *Terminal plan* in the ***Graphical lists*** area in the ***Workspace Explorer***.

2. Right-click with the mouse.

3.CO **Generate**
The ***Select Terminal Row*** dialogue appears.

4.> Choose the terminal strip(s) for which you wish to generate a terminal plan.
By default all terminal strips are selected.

5.> Click **OK**.
The terminal plan has been created.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

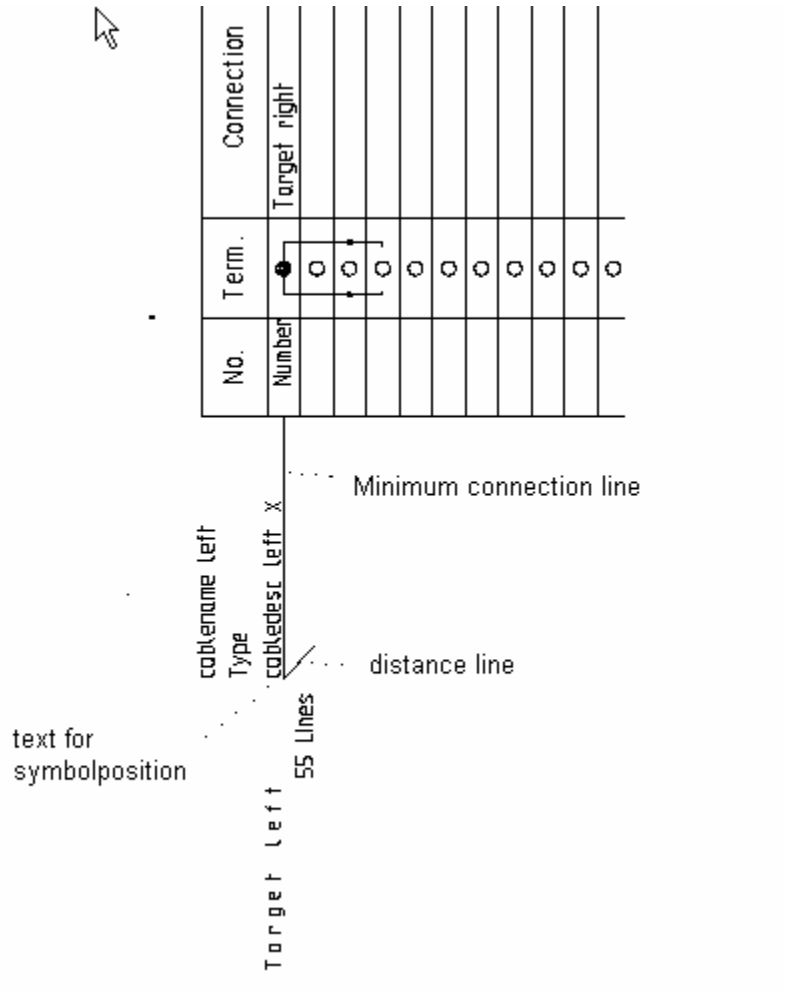
Exercise 32-2: View the terminal plan. Each terminal strip is displayed on a separate page.

1. 0001
   Select page 1 of the terminal plan by double-clicking on 0001 under *Terminal plan* in the ***Workspace Explorer***.
2. 0002
   Select page 2 of the terminal plan by double-clicking on 0002 under *Terminal plan* in the ***Workspace Explorer***.

II.2. CREATING A TEMPLATE FOR A TERMINAL PLAN

Forms for terminal plans represent page templates with special properties. In general, they are created as forms for terminal matrices.

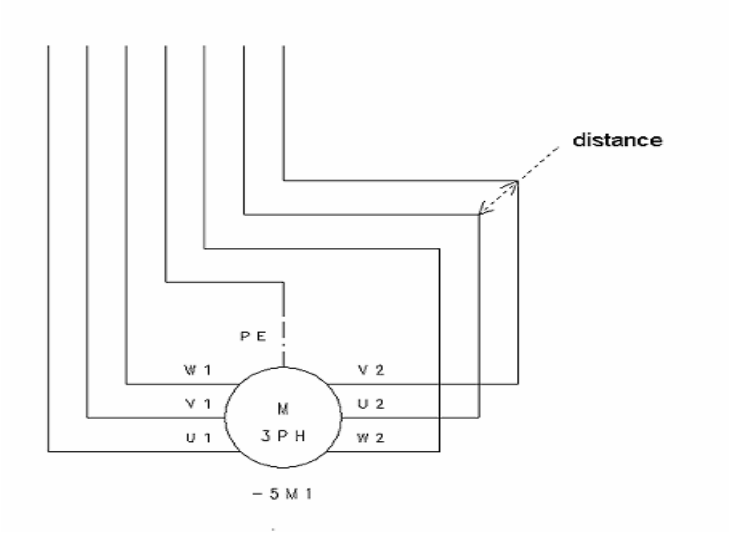
Two "routes" and one "text" are needed in addition, in order to define where the symbols must be placed and how the wires are to be represented.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

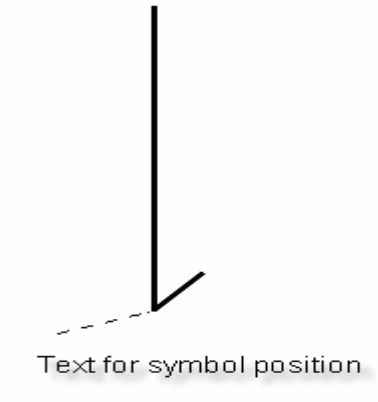
1. **First route: Minimum connection route**
The route must display a connection between the place where the text of the first terminal will appear and the place where the component symbol will appear.
This route will be extended to the first connection of the component symbol while generating the terminal plan, if necessary.
Only vertical or horizontal lines are allowed.

2. **Second route: The route defines the distance between the terminal row and the last remote object**.



If several connection lines exist, this route defines the distance between the connection lines; to be more precise: the difference between the *x* and *y* values defines the distances between the connection extension lines.

3. **The text**
The text defines where the reference point of the component is placed.

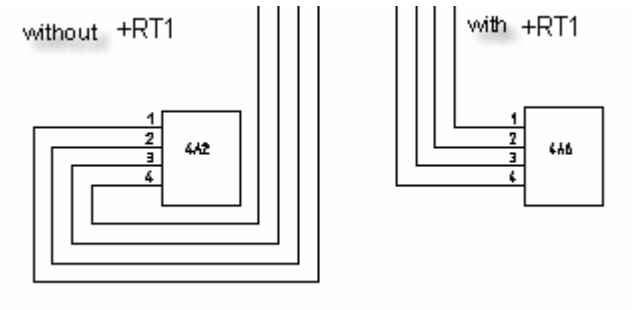


The "routes" and the "text" have special identifications. Insert the Routing symbol from the **List construction set** symbol database into the Terminal plan and change it, if you want to create a completely new Terminal plan.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

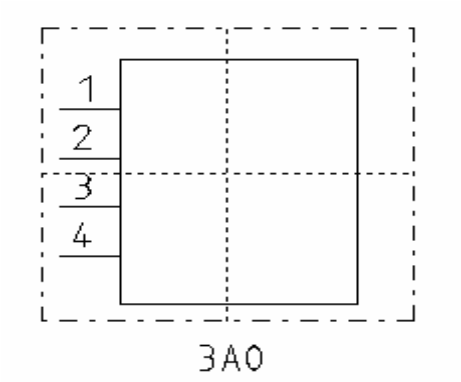
Routing
If the connection of a symbol is not located on the top side of the symbol, the connection lines are drawn on the right side round to the symbol.

If you insert the +RT1 text in the text placeholder for the left target, the connection lines are drawn directly to the symbol connections.



Symbol scaling
+SF, +SX and/or +SY in the text placeholder for the left target defines the Symbol scaling in the Cable plan.
+SF defines a scaling factor in X and Y direction.
Example: +SF0.8 makes all the symbols smaller by the factor 0.8.
+SX or +SY define limits for the component in X/Y direction. *Example*: +SX100: If the extension of a symbol in X direction is greater than 100 mm, then the scaling factor for this special component is defined so that its extension in X direction is less than or equal <= 100 mm. +SY applies adequately.

Explicit component text insertion
The *Function*, *Location*, *Product*, *Description* and *Type* component texts can be displayed at places different from those where they are located within the symbol in the *Circuit diagram*.
The texts are located outside a rectangle that frames the symbol.



You must define the position of the texts in the Form for the *Cable plan* as follows:

- Insert a text with a Function, Location, Component name, Description and/or Type attribute.

If you do not work with Function/Location, you do not need to insert the placeholders. If the Function must be shown at the place where the text is located at the symbol in the Circuit diagram, do not insert this text, etc.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

If the text placeholder for the component name contains the =+- signs, the Function and Location, as well as the Component name appear consecutively in this text.

- Enter one of the following definitions in the placeholder for the left target of the terminals:
  - ✓ +MTX0: place the text on the right to the component
  - ✓ +MTY0: place the text beneath the component

- Group the component texts and the text for the left target as a "*Macro/Group*" symbol.

You cannot use here all of the settings options available in the template. Refer to the "*Graphical lists*" chapter in the Help files for more details.

- Display cable core information and colour, potential name or signal type

The texts with the "Cable-core Number Left" (Id=180128) and "Cable-core Number Right" (Id=180131) attributes are used to display a lot of information.
The first letter in the text allows you to decide which information is shown for a cable-core and the second letter defines which text is shown for the wire:

-	shows nothing
+	shows number (default)
N	shows number (default)
C	shows colour (if colour exists)
c	shows colour (if colour exists) and number (if colour does not exist)
E	shows always colour (also with number not used)

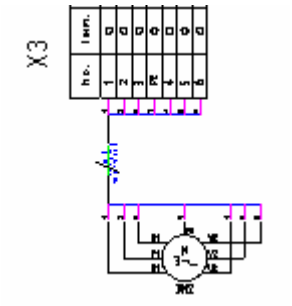
If you need more information, you can position a text with the "Cable-core Number Left" and "Cable-core Number Right" attributes a second and a third time and display the information for signal type or potential name with their help.

Q	generates information about the signal type
P	generates information about the potential name

II.2.1. DRAWING GRAPHICS ASSOCIATED WITH EACH TERMINAL

As described in the chapter "*Creating a Template for the Terminal Matrix*", it is possible to use a symbol for each terminal instead of a grid for the whole page.

II.2.2. DESENHANDO CABOS AS A GROUPING OF CABLE CORES



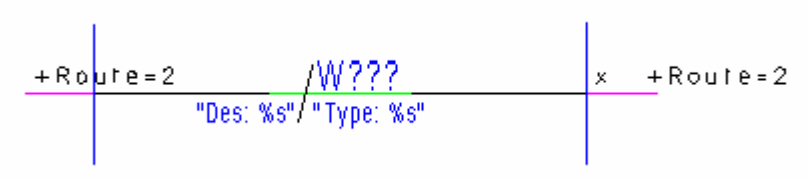
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

The groupings of cable cores are generated because, instead of texts for the cable information, definitions for a cable group are added into the page template for the *Terminal Plan*. The text for the number of lines available for cables has to be defined additionally to this cable group.

Generating the page template:

Generate all objects necessary for the terminal plan as already described. (You can load an existing page template and delete the texts for the cable information from this template.

- Block all this objects as "Block/Macro/Group".
- Generate the cable group as follows:

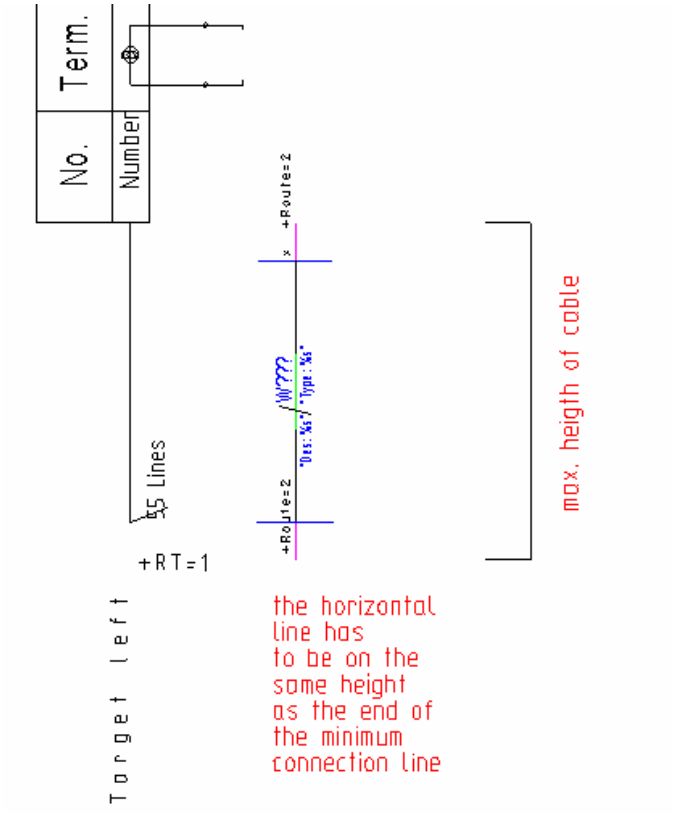


(The symbols are shown rotated by 90 degrees.)

The cable symbol (ID=1600) consists of 3 blocks:

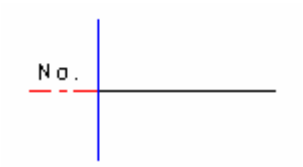
- ✓ *the routing information from cable to target*
- ✓ *the cable*
- ✓ *the routing information from cable to terminals*
- ✓

After you create all 3 parts, block them as "Cable".
The length of the three parts together has to be equal to the distance between the geometry of the terminal plan and the position of the text placeholder "Graphical target left". Additionally the horizontal line in the cable has to be at the same height as the end of the line for the minimum distance.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Part 1: Generate routing-information from cable to the target:



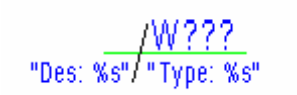
The part for the routing-information from the cable to the target consists of:

- ✓ the text placeholder for cable core number left (ID 180128), for example. No.
- ✓ the text placeholder for cable core square (ID 180138) (optional)
- ✓ the minimum connection line, (black horizontal line, ID 180125)
  As in the terminal plan, the line shows a connection between the positions, where later the text for the first cable core is found, to the position, where the component symbol is found. Eventually the line is extended to the position, where the first contact point of the component is found, when generating the plans.

- ✓ line for cable core number (lilac, horizontal line, ID 1 (shown dash dotted here))
- ✓ vertical line, used to connect all cable cores, ID 1, shown blue here. This line crosses the other two lines in their common endpoint.

  - Block all these objects as "Block/Macro/Group".

Part 2: Generate cable:



The part for the cable contains:

- ✓ The geometry for the cable.
  Generate the geometry like you normally do. You can add normal texts if you want. Block all as "*Graphical Symbol*".

- ✓ Text placeholder for cable name, for example W??? (ID 180110)
- ✓ Text placeholder for "Type" (ID 180140)
  Example: Type %s" where the placeholder "%s" is replaced by the cable type.

- ✓ Text placeholder for "Description" (ID 180142)
  Example: Des: %s" where the placeholder "%s" is replaced by the cable description.

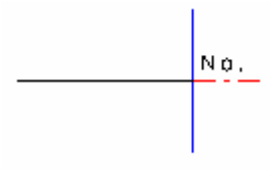
- ✓ Text placeholder for "cable type" (ID 180144)
  Example: Dim: %s" where the placeholder "%s" is replaced by the text "cable-type".

- ✓ Text placeholder for "Length" (ID 180145)
  Example: L= %s" where the placeholder "%s" is replaced by the length

  - Block all these objects as "Block/Macro/Group".

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

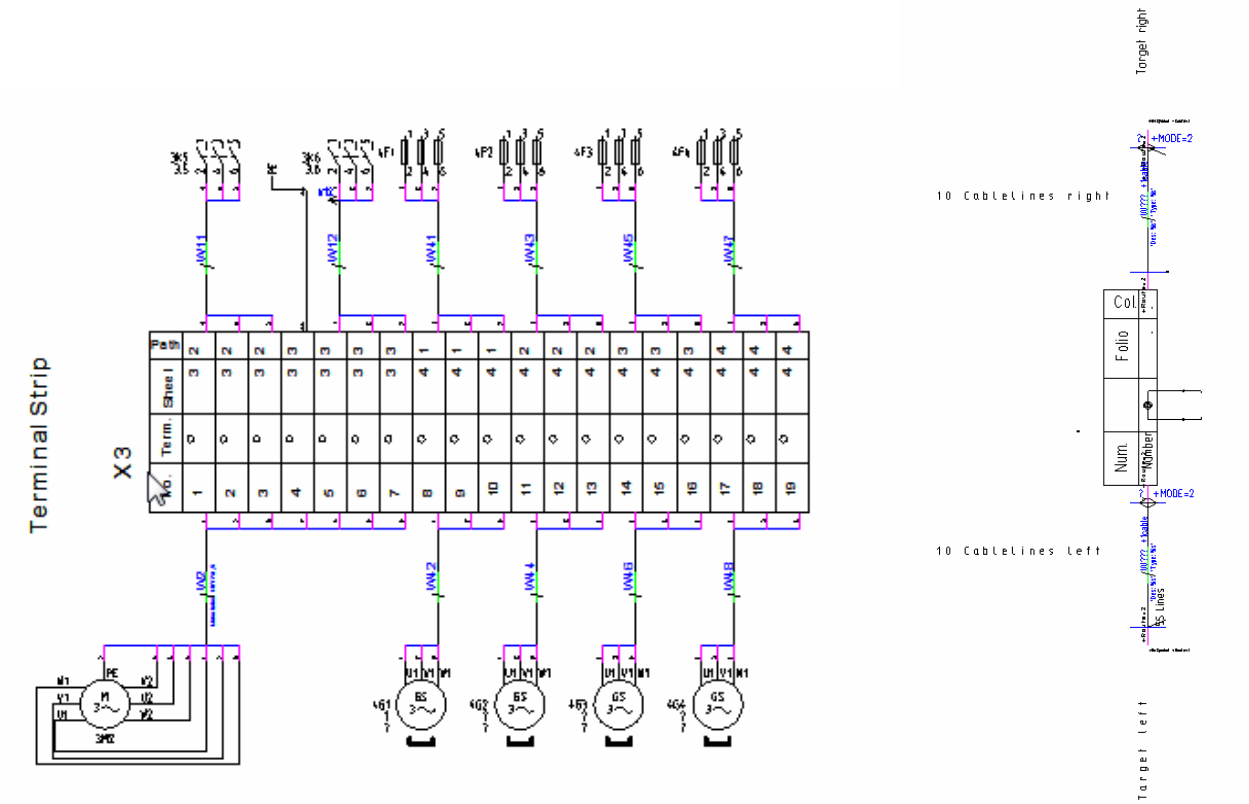
Part 3: Generate routing-information from cable to terminal



This block consists of the same elements as the cable core left mirrored.

- Block all these objects as "*Block/Macro/Group*".
- Select the blocks "Routing-information from cable to target", "Cable" and "Routing-information from cable to terminal".
- Block all these objects as "*Block/Macro/Group*".
- Position the number of lines on top of the minimum connection line.
- Block all the objects as "*Page Template, Title block*".

II.2.3. DESENHANDO CABOS/GEOMETRY ON BOTH SIDES OF THE TERMINALS



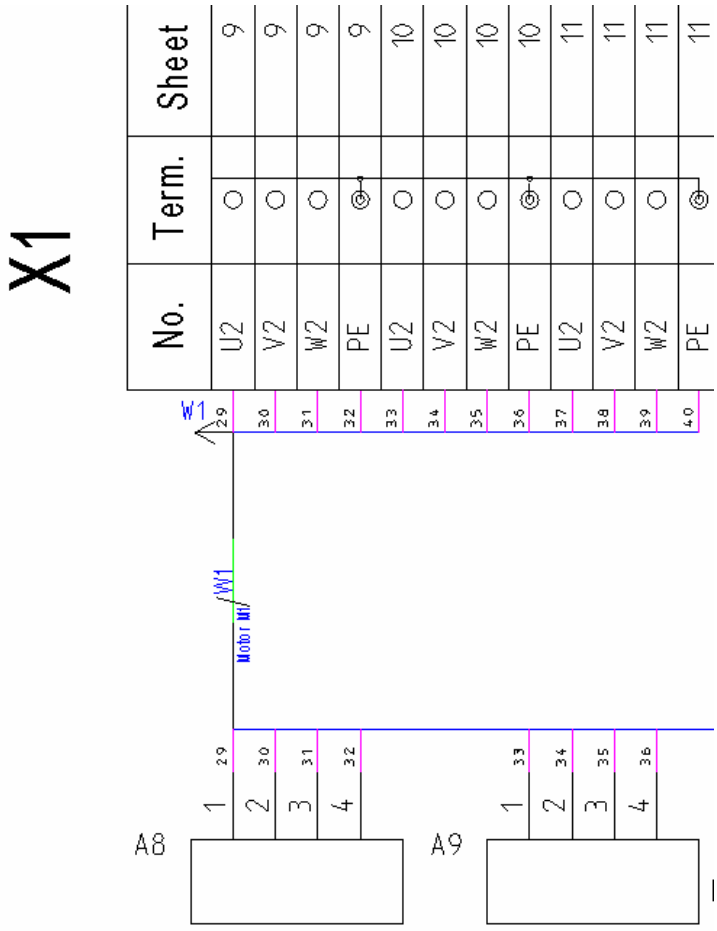
If you want to add the graphical representation to both sides of the terminal, you can insert the same objects used to insert the representation on the left side, just use "Target right " instead of " Target left".
Of course, you also have to use " cable core number right" instead of "cable core number left" etc for the other necessary texts. One text is orientated in the direction of the connection point of the terminal and the other is orientated in the direction of the connection point of the target side.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

II.2.4. SHOWING THAT CABLE CORES ARE USED ON PREVIOUS/NEXT PAGE

The *Terminal Plan* can generate information showing whether cable cores of a cable are used on previous W1 or next W1 page.

Example:



In the page template you have to add two macro/groups to each placeholder for left or right cable:

- ✓ group for cable reference symbol to previous page;
- ✓ group for cable reference symbol to next page.

Each cable reference group contains the following elements:

- ✓ text with attribute "cable reference LAST" or "cable reference NEXT" (depends on which cable reference group you are preparing)
- ✓ geometry which represents the cross reference, for example .

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

With the text with the attribute "cable reference LAST" or "cable reference NEXT" you can control the position on which the cross-reference symbol appears. Use the following keywords to do so:

- ✓ +Mode=0 (and other values): the Cable reference-symbol is not drawn
- ✓ +Mode=1: the reference-symbol is moved to the end of the grouping-line of the target-routing;
  If the collecting-line of the target-routing gets an offset, then both cable reference symbols get the same offset.
- ✓ +Mode=2: the reference-symbol -symbol is moved to the end of grouping-line of the terminal-routing;
  If the collection-line of the terminal-routing gets an offset then both cable reference symbols set the same offset.
- ✓ +Mode=3: the Cable reference symbol is not moved (relative to the cable name).

In the template the following flags control the behaviour:
If the cable-name left (id=180127) and cable-name right (id=180130) in the template each contain the substring "+1Cable", then the cable name is shown only once at each terminal row (each graphical cable has only one core-grouping).

Cable-core-routing:
The macro/group for the left cable routing contains 2 core-groupings, which each contain a "Cable-core Number Left" (id=180128) (together with 3 lines building the "cross").
If the "Cable-core Number Left" contains the text "+Route=1" or "+Route=2", the cable-core routing style is changed as follows:

- ✓ +Route=1 (Default): each core-grouping line automatically gets an offset (this is safer, because lines do not appear that easily on top of each other).
- ✓ +Route=2: If this setting is found, the core-grouping line gets an offset only if it intersects with another core-grouping line.

The symbol for the right cable routing works in the same way, but uses "Cable-core Number Right" (id=180131).

Note :

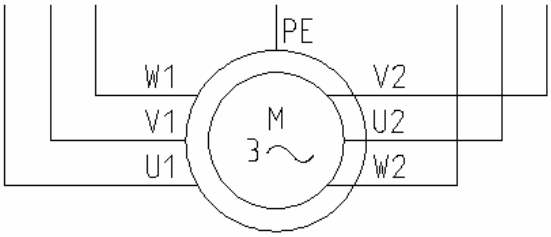
If a cable is on the left and right side of the terminal-row (only one page), the Cable-reference-symbols may not be correct.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

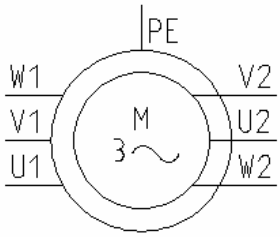
II.2.5. SYMBOLS FOR A TERMINAL PLAN

At the terminal plan generation, the component symbols used in the circuit diagram are placed. Drawing connecting lines in the terminal plan is better possible if the connections of the components are stuck out upward:

This way:



Not this way:



Suppress symbols:
If you add the +NoSymbol text to the left target, no symbols are drawn, only the component text is used as target.

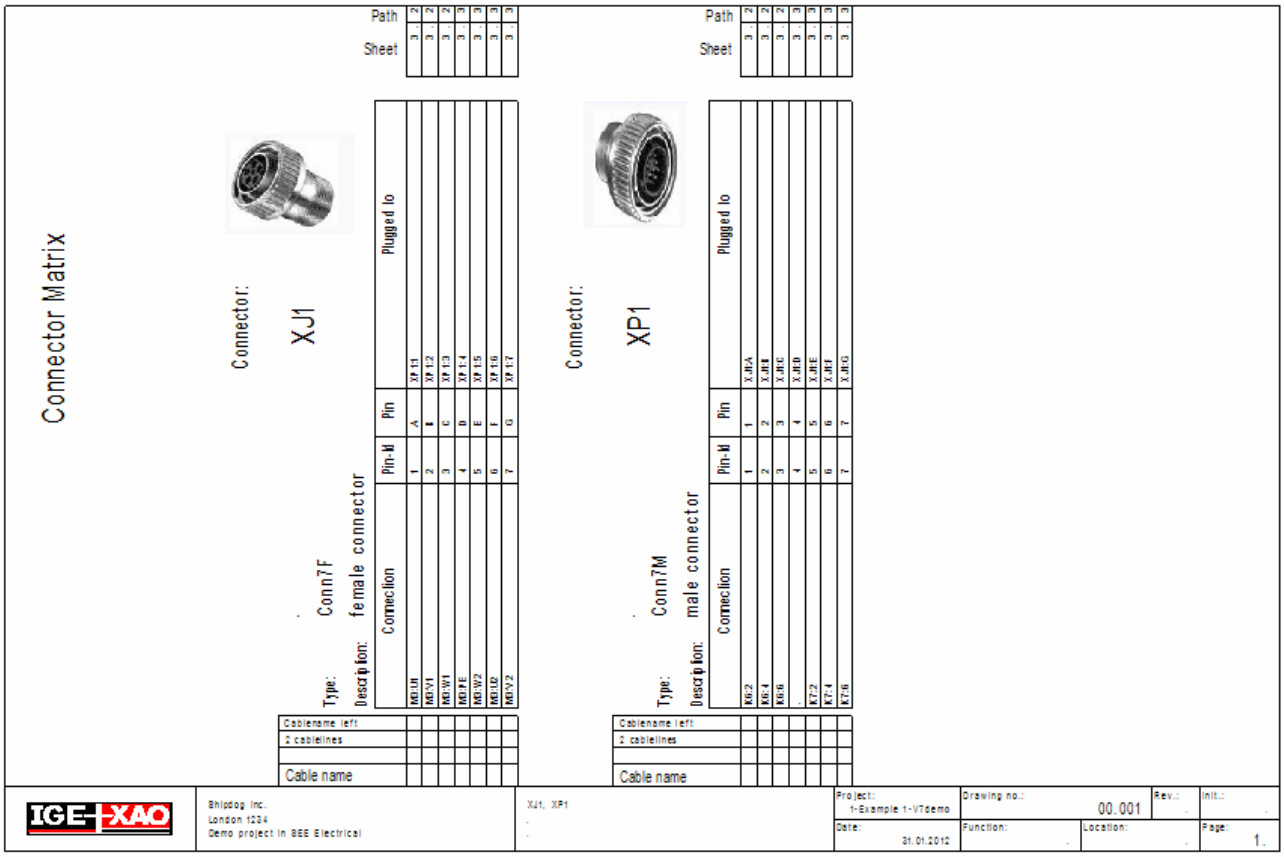
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

JJ CONNECTOR MATRIX AND PLAN

(Advanced)

JJ.1. GENERATING A CONNECTOR MATRIX

The connector matrix facilitates the installation of connectors.



Exercise: Generate a connector matrix.

1. Select *Connector matrix* from within the *Graphical Lists* in the Workspace tree. Expand the *Graphical Lists*.

2. Right-click with the mouse.

3.CO **Generate**
In the ***Select Terminal Row*** dialogue, choose the connector(s) for which you wish to generate a matrix, for example X2 and X3.

4.> X2, X3

5.> OK
The Connector matrices have been created.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

JJ.2. CREATING A TEMPLATE FOR A CONNECTOR MATRIX

Templates for connector matrices are page templates with special properties.

A *Connector Matrix* provides similar possibilities, which you know from *Terminal Matrix*.
That is why, the template creation follows similar rules, too.

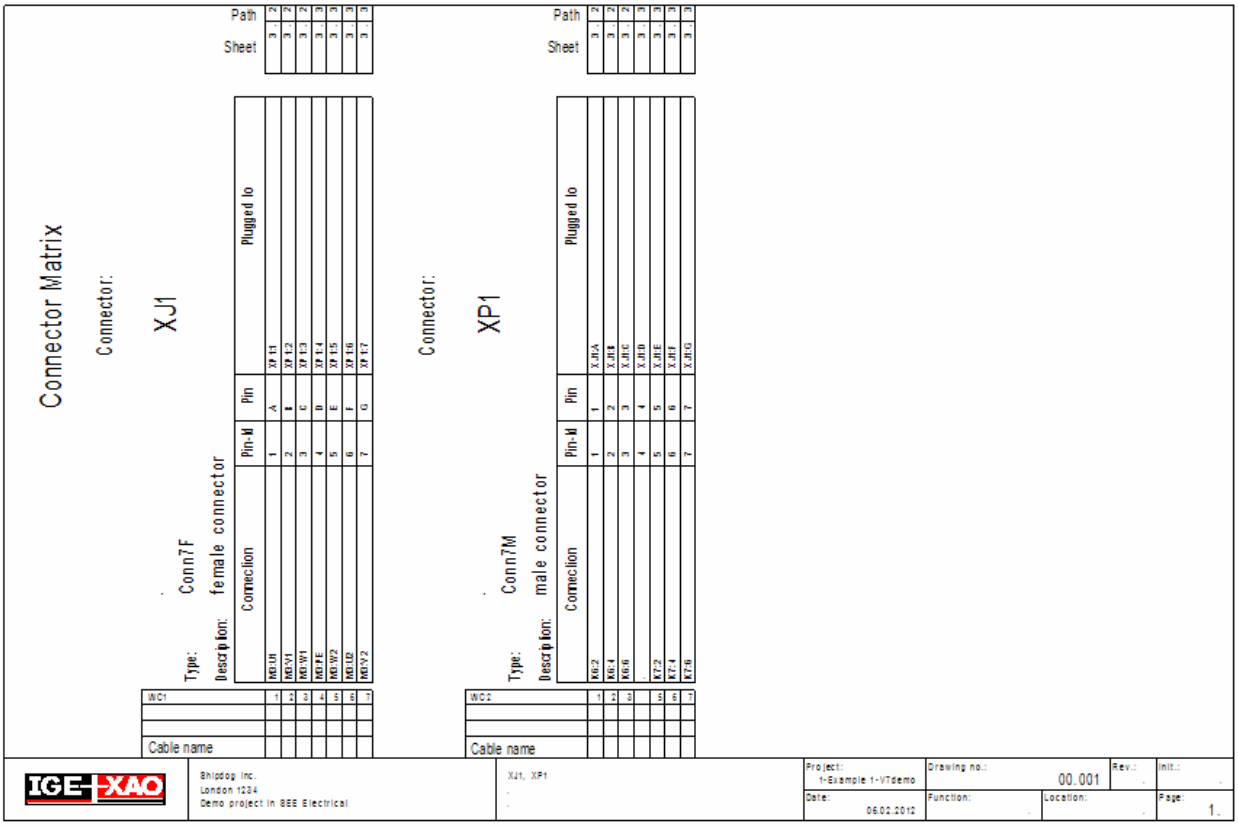
Differences in the text place holders between Terminal Matrix and Connector Matrix (or Terminal Row Picture and Connector Plan):

	Terminal Matrix and Terminal Row Picture	Connector Matrix and Connector Plan
Header: (ID=180110)	Terminal strip	Connector
Row:	Terminal	Pin
ID=180112	Terminal number	Connector Pin
ID=180137	Terminal index	Pin-ID

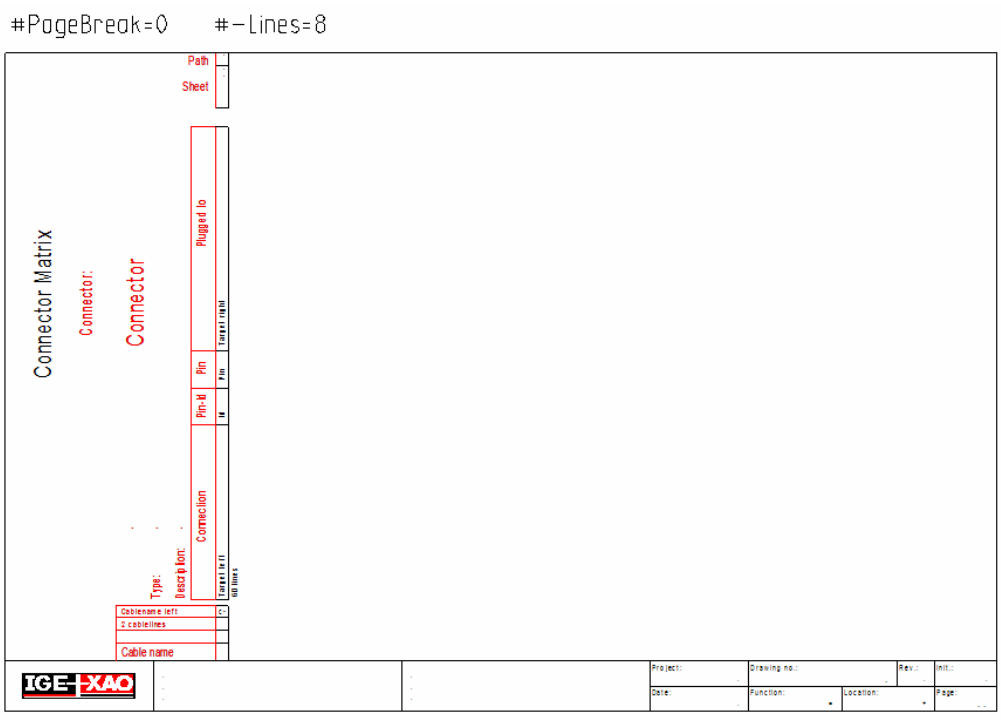
In a Connector Matrix or a Connector Plan, cables are entered only on the left side of a pin because the plug on its second side will be always plugged into another connector. By default, the attached connector is always shown on the right side of a pin, so cables are not necessary here.
Bridges are not required in the Connector Matrix or in the Connector plan, so there are no placeholders in the page template.
More information about this you can find in the Manual.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

JJ.3. DRAW MORE THAN ONE CONNECTOR PER PAGE



Generating the Page template:



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

1. Group the geometry and texts from the title block as "Page Template/Title Block" symbol.
2. Generate all geometry and texts that show information that belongs to the Connector.
3. Place a text with the attribute "Connector".
4. Generate all geometry and texts that show information that belongs to the pin. The geometry and texts that are not changed have to be grouped as "Graphical symbol".
5. Place a text with the attribute "Pin number".
6. The graphical symbol and the text with the attribute "Pin number" have to be grouped as "Macro/Group".
7. . Add a "Normal" text attribute to the content.

#PageBreak=0
or
#PageBreak=0 #Lines=8 (for example)
where #Lines=8 indicates that multiple connectors will be generated and #Lines=8 indicates that the "connector header" uses the space given for 8 terminal-lines (if no #Lines is used to specify the connector header, the necessary space is calculated from the graphic found in the template)

The text "#Pagebreak" in the template can be used to control the generation of the new page.
#PageBreak=0
Multiple connectors are generated on one page like before, a new page is generated only if the page is full.

#PageBreak=1 or #PageBreak=160010 or #PageBreak="-"
are used to start a new page f the name of the connector has been changed.
#PageBreak=140020 or #PageBreak="="
are used to start a new page if the function (=) has been changed.
#PageBreak=140050 or #PageBreak="+"

Are used to start a new page if the function (+) has been changed.

You can define how is managed the function and location information in the connector's name.
The connector's name is always extended with the function and location information if you add the text +DL0 to the #PageBreak command (even if the connector has the same function and location information then the page it is found on).

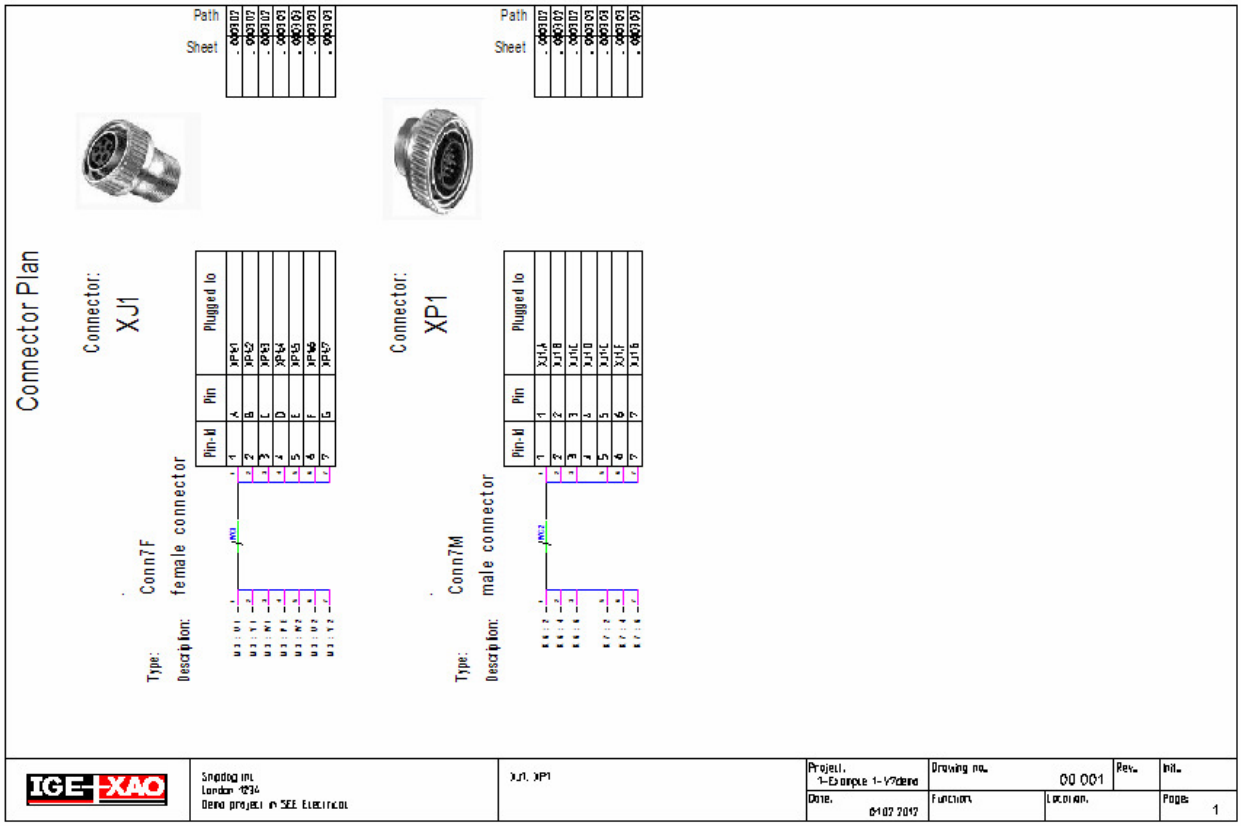
If you add the text +DL1 to the #PageBreak command then the connector's name contains only the function and location information in case they are different from the ones found on the page.

8. Group all symbols (the macro/group for the standard sheet and the macro/group for the single terminals) as a "Page Template, Title Block" symbol.
9. Save the new page template.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

JJ.4. CONNECTOR PLAN

(Advanced)



The *Connector plan* is created following the same rules as the *Terminal Row Picture*, but of course, it shows connectors and their pins.
More information about creating of templates you can find in the Manual.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

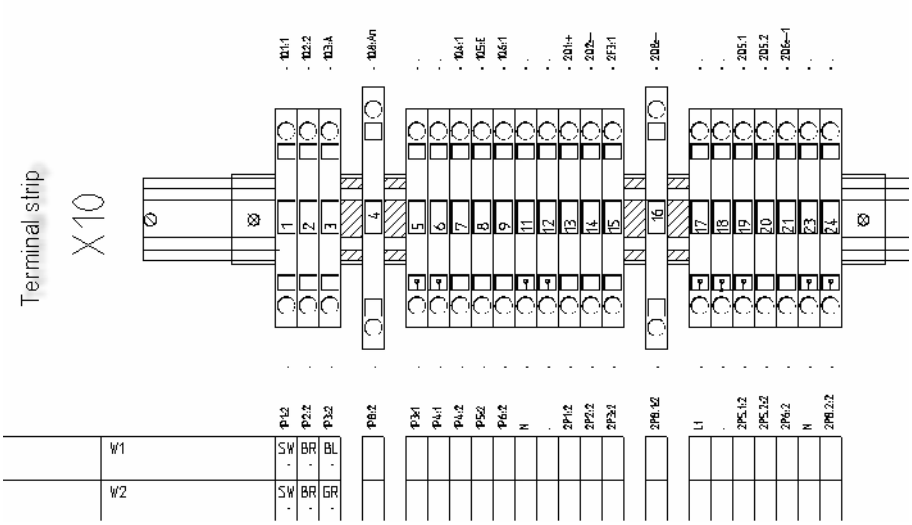
KK TERMINAL ROW PICTURE PLAN

(Advanced)

KK.1.GENERATING A TERMINAL ROW PICTURE PLAN

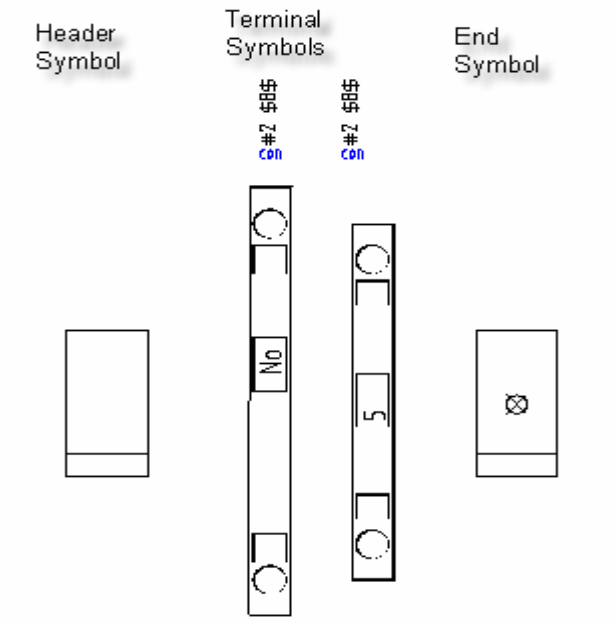
The *Terminal Row Picture Plan* enables you to insert a specific symbol for each terminal. For example, if single terminals in a terminal strip are mixed with multi-layer terminals, you can see directly which these are. In addition, a diode terminal can be displayed in a different way than a Switching terminal, etc.

Example:



It is possible, by means of a Header, to show additional information in front of the terminal as the one shown above the cover plate of the terminal strip. The End symbol accomplishes the same.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



You can specify in three ways which symbol must be used for displaying the terminal:

Via the terminal type:

In the *Type database*, you can assign the symbol by selecting the "Symbol name for terminal plan (multi level)" property or select the symbol by means of the ***Symbol*** browser.

You can also determine, in addition to the terminal symbol, which Header symbol and which End symbol to be used in the Terminal plan (multi level). The definition must be formed as follows: +T="<Terminal Symbol Name > +H="<Header Symbol Name>" +E="<End Symbol Name>".

If you do not wish to place a "*Header Symbol*", you can type +H0 behind the name of the terminal symbol for the *Terminal plan* (multi level), therefore +T="<Name>" +H0.
If you wish to give, before each terminal, the "*Header Symbol*" indicated in the Symbol for the *Terminal plan* (multi level), type +H1, therefore +T="<Name>" +H1.
If you wish to give, behind each terminal, the "*End symbol*" indicated in the Symbol for the *Terminal row picture*, type +E, therefore +T="<Name>" +E.

Via the terminal symbol in the Circuit diagram:

You can define a symbol by the "*Symbol name terminal plan*" text. The symbol is searched in the TERMINALROWPICTURE symbol library of the "*Terminalplan*" symbol folder. If it must be searched in another location, please specify <Symbol database>\<Folder>\<Symbol name>.
This symbol is also used if another symbol is assigned by the type.

You can also determine, in addition to the terminal symbol, which "*Header Symbol*"and which "*End symbol*" must be used in the Terminal plan (multi level). The definition must be formed as follows: +T="<Terminal symbol Name> +H="<Header Symbol Name>" +E="< End Symbol Name >". (If you wish to insert Header and End Symbols here, you must always indicate the Symbol for the Terminal Row Picture plan, too!)

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

If you do not wish to place a Header Symbol, you can type +H0 behind the name of the terminal symbol for the Terminal Row Picture plan, therefore +T="<Name>" +H0.
If you wish to give, in front of each terminal, the Header Symbol indicated in the Symbol for the Terminal Row Picture plan, type +H1, therefore +T="<Name>" +H1.
If you wish to give, behind each terminal, the End Symbol indicated in the Symbol for the *Terminal Row Picture* plan, type +E, therefore +T="<Name>" +E.

If you want to assign, behind the last symbol of a terminal strip, the "*End Symbol*" defined in the Symbol for the *Terminal Row Picture* plan, and if the symbol for the *Terminal Row Picture* plan is given in the Type, then fill in here, in the Circuit diagram, only an +E for this symbol.
In this way, you can assign the symbols for the Terminal row picture to the terminal types and then set the end plate flexible via +E at the symbol, or accomplish varying definitions.
Default Symbol

If a symbol name is found neither in the terminal symbol nor in the type, the "0Terminal" symbol from the *TERMINALROWPICTURE* Symbol database is inserted.

Exercise 33-1: Generate the Terminal Row Picture Plan for the training workspace.

1. Select *Terminal Row Picture Plan* in the ***Graphical lists*** area of the ***Workspace Explorer***.
2. Right-click with the mouse.
3.CO **Generate**
The ***Select Terminal Row*** dialogue appears.

4.> Choose the terminal strip(s) for which you wish to generate a terminal row picture plan. By default all terminal strips are selected.
5.> Click **OK**.
The *Terminal Row Picture Plan* is generated.

Exercise 33-2: View the Terminal Row Picture Plans. A separate sheet is created for each terminal strip.

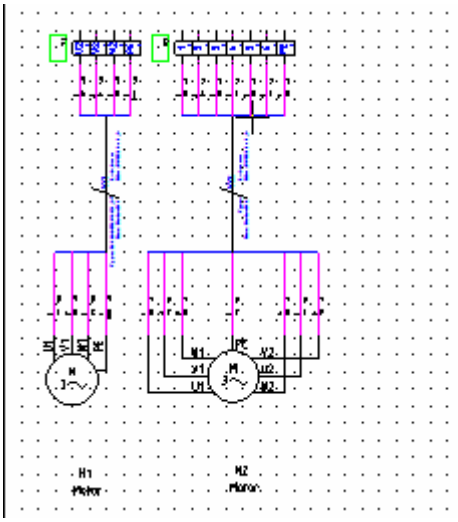
1. 0001
Select sheet 1 of the *Terminal Row Picture Plan* by double-clicking on 0001 beneath **Terminal Plan (multi level)** in the *Workspace* tree.
2. 0002
Select sheet 2 of the Terminal Row Picture Plan by double-clicking on 0002 beneath **Terminal Row Picture Plan** in the ***Workspace Explorer***.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

LL CABLE TERMINAL ROW PLAN

(Advanced)

The *Cable Terminal Row Plan* is a plan which allows you to generate various views of the cables and the terminals in the circuit diagram, according to the selected template.



For further details on how to generate a page template for the *Cable Terminal Row Plan*, please consult the "*Graphical Lists"* chapter of the Help files.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

MM PRODUCT ASSEMBLY LIST

(Advanced)

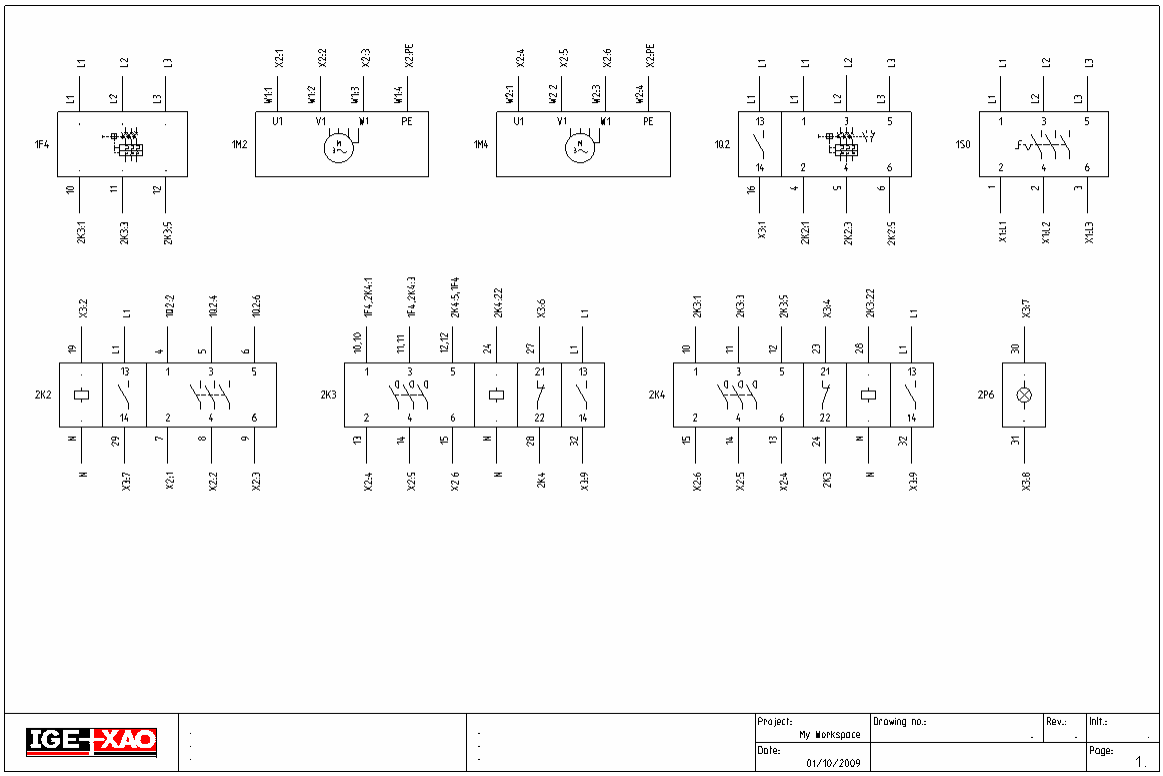
This list gives a complete overview of the entire component by collecting all master and slaves information and presenting all information in one symbol.

The *Product Assembly* list is generated using the same symbols for the components as used in the circuit diagrams. A rectangle is created automatically around the symbol.

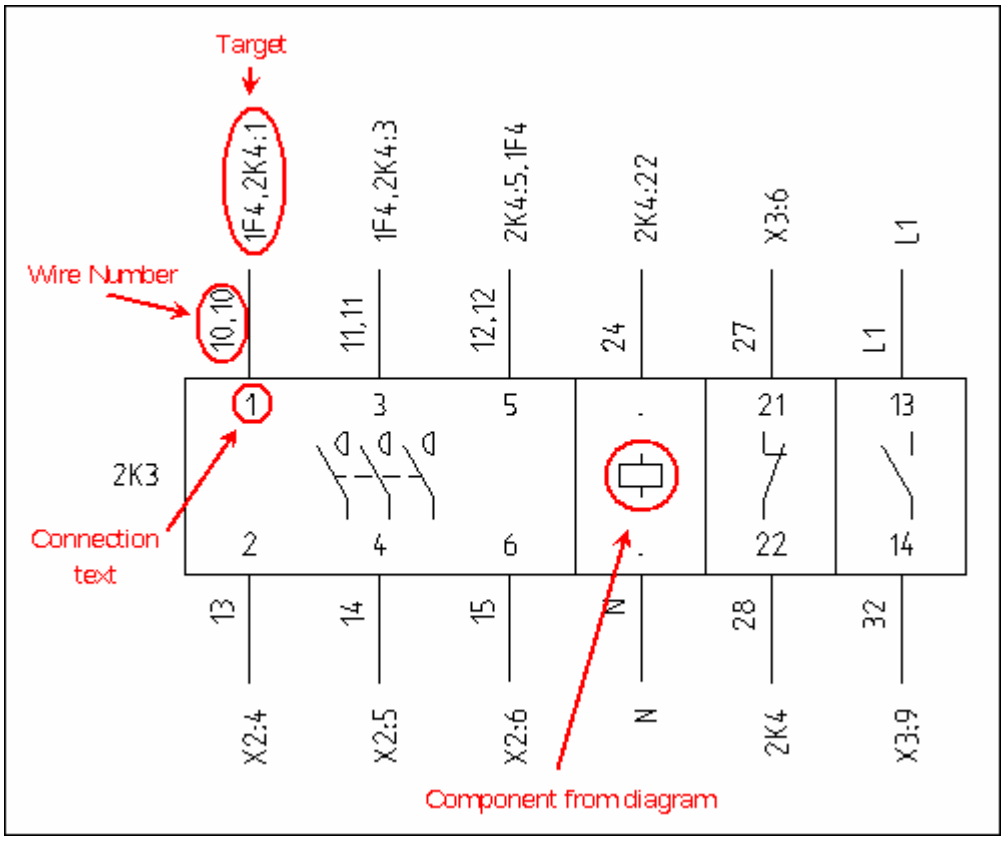
For the components which consist of master and slave symbols, like relays for example, the master and slave symbols are grouped together in a common rectangle. The sorting order inside this group depends on the sort order set in the channel definition for the assigned type. Items not present in the diagram are also shown but with empty wiring information.
If no channel information exists, the individual items are arranged in the order in which they are inserted into the diagram.

If wire numbers are defined in the circuit diagram, they will appear on the wires in the assembly list as well.

Example:



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

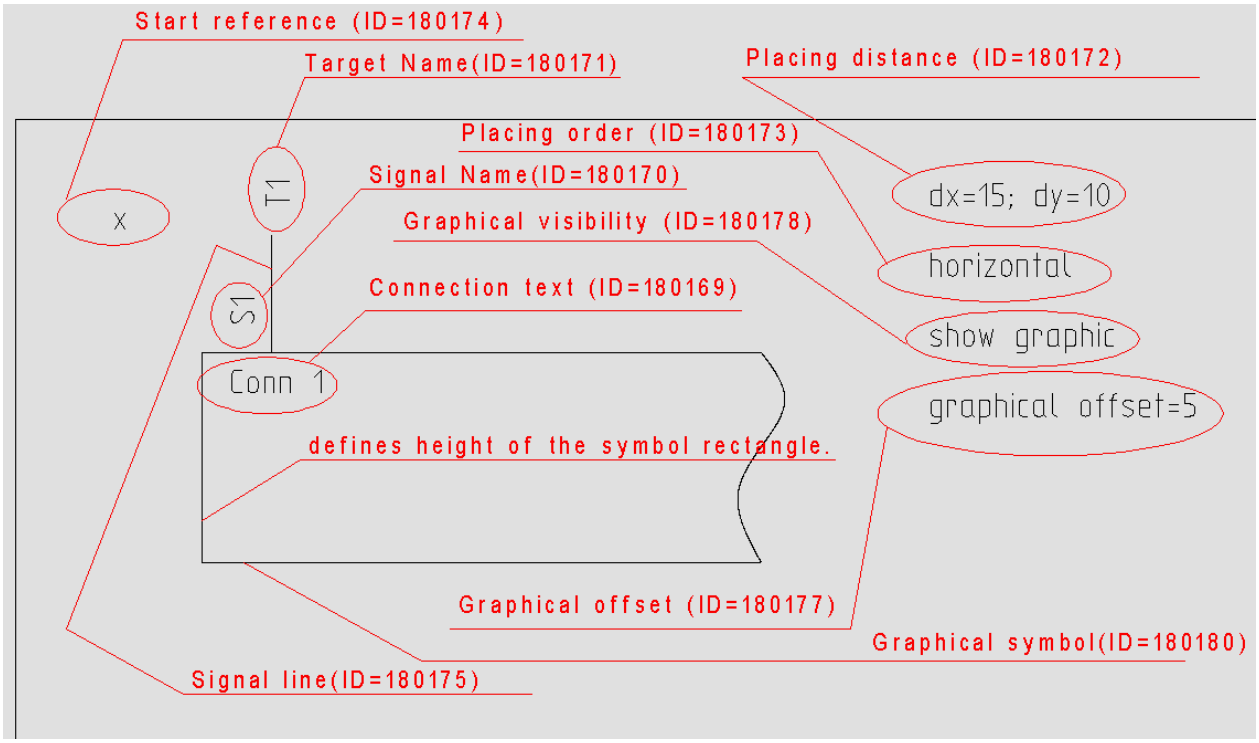


The page template controls the distance, the orientation and the layout of the generated graphical list.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

MM.1. GENERATING A TEMPLATE FOR THE PRODUCT ASSEMBLY LIST

Besides the ordinary placeholder texts in the page template, the following special placeholder texts are necessary:



You find all the different text attributes in the "***Other***" area in the text attributes list.

- Add a text with the attribute "*Start reference*" (ID 180174), for example "X".

This text defines the start point for inserting the component symbols. It also controls the area used for placing the components.
"ex=420; ey=50" or "endx=420; endy=50" defines the (absolute) x and y value of the right bottom edge.
"dx=410; dy=205" defines the area to the reference point of the "Start reference" text.

Note :

*You get exactly the same component if the right / bottom edge is on the wrong side of the "start reference" text (no rectangle inside the drawing boundaries). If no right edge for the placing is defined in the template, the right bottom edge is calculated from the "X- and Y- Extension of Page" properties: right side of drawing area = (max. x-coordinate of drawing) - 2*0.03* (max. x-coordinate of drawing) and bottom side of drawing area = (min. y-coordinate of drawing) * 0.101*

- Add a text with the attribute "*Placing distance*" (ID 180172), for example dx=15; dy=10.

This text defines the distance between two components in x- and y– direction.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

- Add a text with the attribute "*Placing order*" (ID 180173), for example "*horizontal*".

This text defines the order of placing of the components. The key words can be "*horizontal*" or "*vertical*". If it is "*horizontal*", the placing of components starts from left and continues to the right, until the row is full. Then the placing continues in the next row. "*Horizontal*" is the default value.
If "*vertical*" is chosen, the placing of components starts on top and goes to the bottom until the column is full. Then the positioning continues in the next column.

- Add a text with the attribute "*Graphical visibility*" (ID=180178), for example "*show graphic*".

This text determines if the graphical symbols are shown inside every component or not. In order for the graphical symbol to be displayed, the content of this placeholder has to be "*show graphic*". If this is not the case, the graphical symbol will not be shown.

- Add a text with the attribute "*Graphical offset*" (ID=180177), for example "*graphical offset=5*".

This text defines the offset from the boundary rectangle to the graphical symbol. The software looks for the "=" sign and takes the value after it to generate the offset.

- Add texts to control the position of the component name, description and type

If no definitions for component name, function and location are made, the texts appear at their default positions.

- Add a text with attribute "product (-)" to the block for the component inside the template if you want to define the position of the component name or the look of the text yourselves (the attribute is found in the *Component* section in the ***Text Attributes***).

You have to define the position of the component name if you want to define description, type and / or function / location.

- Add a text with attribute "description 00" to the block for the component inside the template (the attribute is found in the *Component* section in the ***Text Attributes***)

- Add a text with attribute "type" (the attribute is found in the *Component* section in the ***Text Attributes***). If more than one type is defined in the component, all types will be shown like type1; type2, etc.

- Add texts with attributes "function" and "location" (attribute is found in the *Function & Location* section in the ***Text Attributes***)

- Add a text with the attribute "*Connection text*" (ID=180169), for example "*conn 1*".

This text defines the attributes for the connection point name and its position inside the symbol.

- Add a text with the attribute "*Signal Name*" (ID=180170), for example "*S1*".

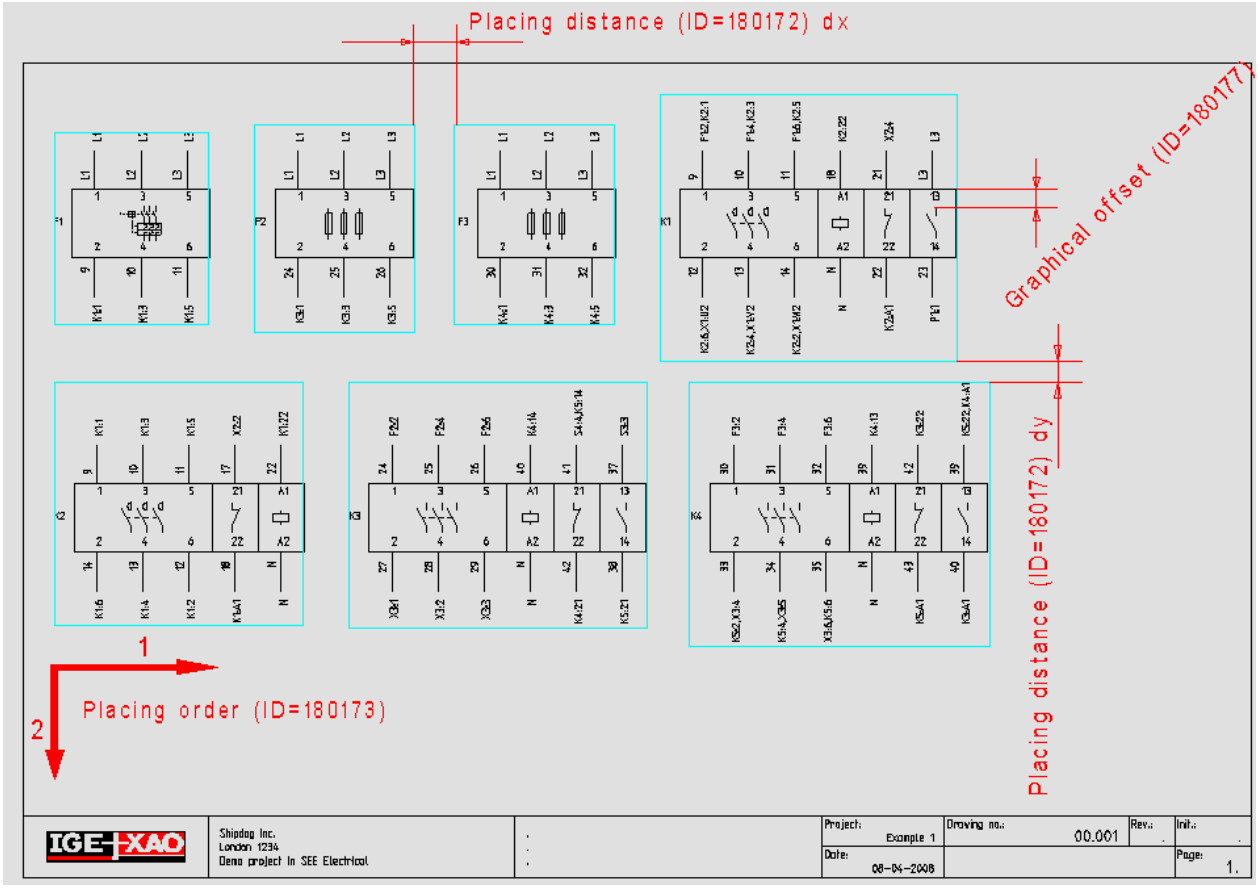
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

This text defines the attributes for the signal name and its position inside the symbol. It can be used to add more information about the wire. The value of the text defines the information which is shown. You can position it max 6 times in the template and use the following codes:

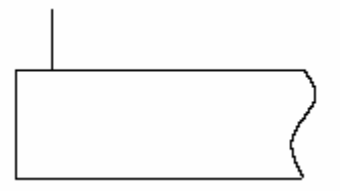
- ✓ *"+N" --> Cable-core or wire number (default flag)*
- ✓ *"+C" --> Cable-core or wire colour*
- ✓ *"+c" --> Cable-core or wire colour and number if colour is empty*
- ✓ *"+S" --> Cable-core or wire size*
- ✓ *"+Q" --> Cable-core or wire signal*
- ✓ *"+P" --> Cable-core or wire potential*

- Add a text with the attribute "*Target Name*" (ID=180171), for example "*T1*".

This text defines the attributes for the target name and its position inside the symbol.

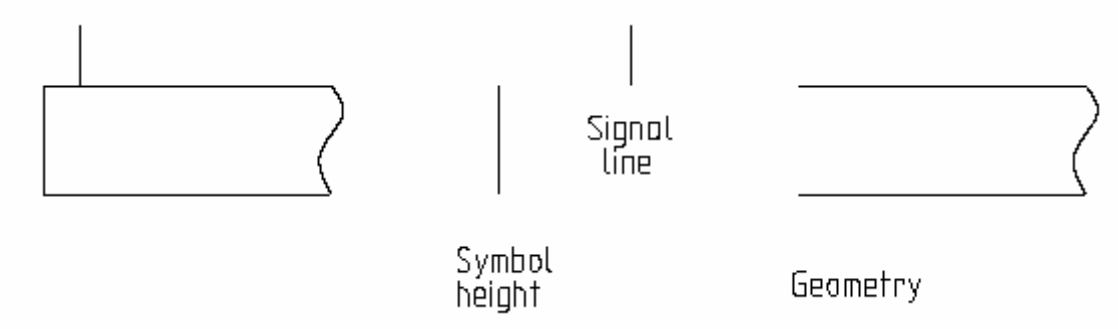


Graphical symbol



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

The graphical symbol contains 3 parts:



- Draw a line (ID 1) and block it as "*Signal line*".

The line has to be strictly vertical. This line defines the length of the connection lines. (If in the parent group of this line there are other elements, they will be placed in the symbol, too.)

- Draw a line (ID 1) and block it as "*Symbol height*".

The line has to be strictly vertical. The line defines the height of the symbol.

- Draw geometry by using the **Draw** □ **Elements** □ **Line** command (part of a rectangle).

This geometry is used to generate the rectangle for the components.

- Block the 3 parts as "*Block, Macro, Group*".

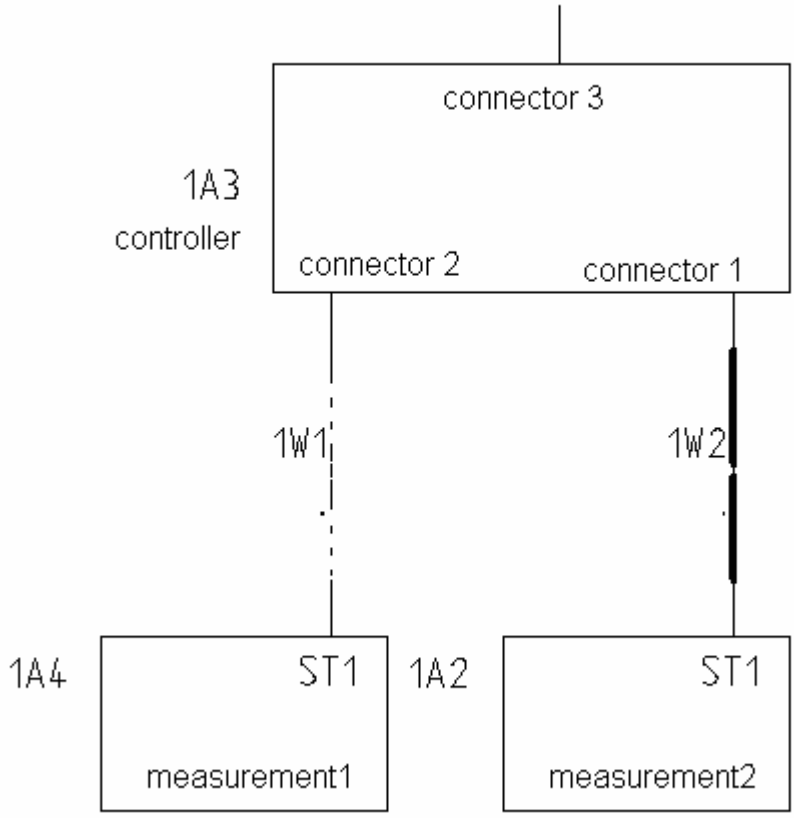
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

NN MULTICORES

(Advanced)

NN.1. USING MULTICORES

Multicores are used for drawing a wire that must contain multiple cables, for example, if you have to go from one circuit board to another circuit board upon connectors.



Multicores can be displayed in the special database lists Multicores and Multicore-wires

Exercise 36-1: Define a wire as a multicore wire.

1.CA **Electrical**
2.CO **Multicore** (**Multicore** panel)
3.+ Identify the wire you wish to define as a multicore.
The multicore receives a name.
You can define another wire as a multicore. Do not do this now. Right-click to exit the multicore drawing mode.

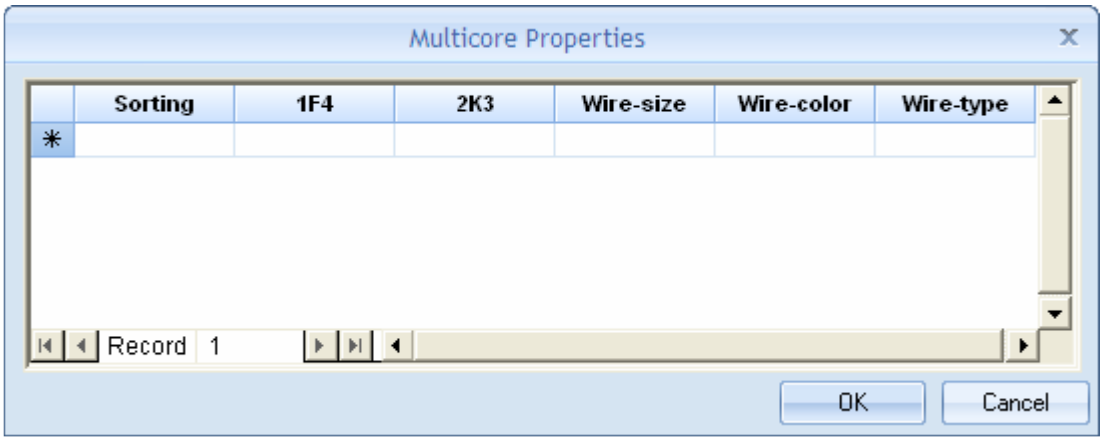
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

4. Double-click the text of the Multicore name. You can enter a description, type and length of the multicores. You will find the corresponding data in the list of multicores afterwards.

5. You can change the pen style and pen width, if you wish. In this way, you can highlight the multicores graphically. Use the functions already known.

Exercise 36-2: Assign wires to the multicores.

1.CA **Electrical**
2.CO **Properties** (**Multicore** panel)
3.+ Select the multicores that you wish to assign single wires to and edit the wires.
The following dialogue box appears:



4.> Click in the line to add wires.
Define the wires sorting, type in the connection of the first component, the connection of the second component, wire-colour, wire-size and wire-type.
To delete a wire, select the line and press the **Delete** key.
5.> After you have defined all the wires in the multicore, click **OK** to close the dialogue box.

Exercise 36-3: View the Multicores and Multicores-wires lists within the Graphical lists.

1. Select **Multicores** in the **Graphical lists** area of the Workspace Explorer.
2. Right-click with the mouse
3.CO **Generate**
The Multicores graphical list is generated.
Double-click the page 001 in the *Workspace Explorer* to open it.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

OO INSERTING COMPONENTS NOT IN THE DRAWING VIA THE DATABASE EDITORS

(Advanced)

You can insert components for the Part list or spare terminals via the database editors. It is not necessary any more to insert them into the diagrams. You can also import such components without graphic with the help of an Excel file.

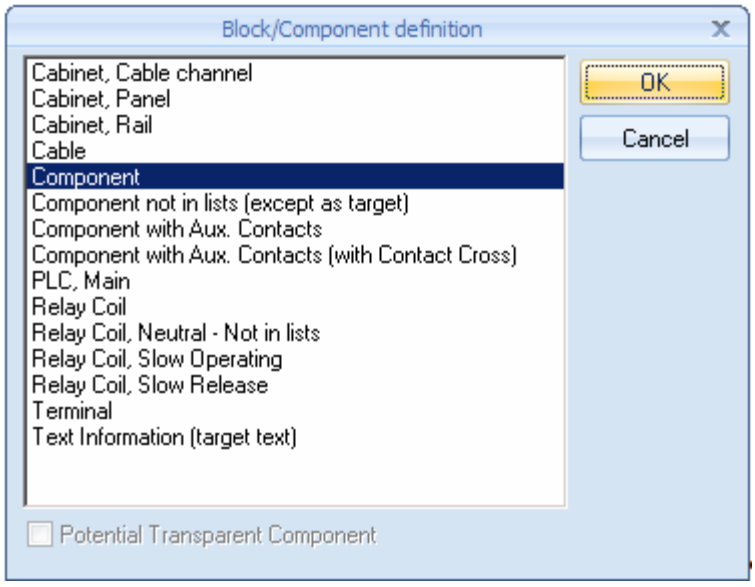
OO.1. EDITOR FOR COMPONENTS WITHOUT GRAPHICS

This Editor contains information about components without graphics in the workspace (for example, spare terminals, separation and end plates for terminals, inserted via the Terminal Editor; a box for shipping of the designed machine, added via the Component Without Graphic Editor, etc.).
It allows you to insert manual components that do not have any graphical representation and are not inserted in the circuit diagram pages.

Quick guide: Inserting components without graphics

1. Open the Component without graphic Editor.

2. Right click and select the **Add new component** command.
The following dialogue appears:



3. Select the desired type for the element without graphics and click **OK**.
The element is inserted in the database list, even if it does not exist in the circuit diagram.
You can modify its properties in the right pane of the Editor.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

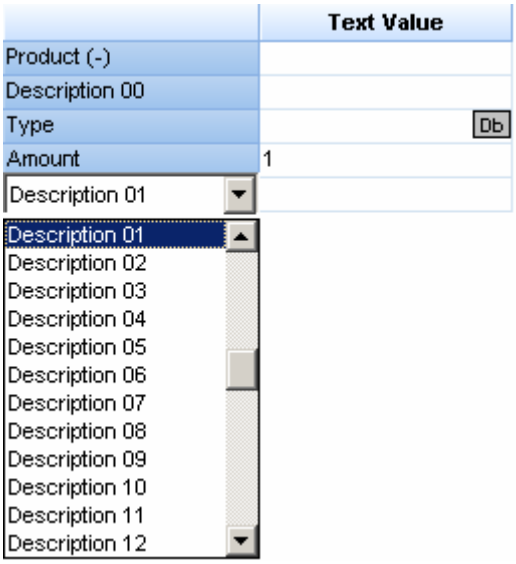
Hint

*If you right-click on a row containing a component and you execute the **Add new component** command, the new component will be a copy of the existing one. To create a different kind of component, right-click the headline of the Component Without Graphic Editor.*

In the right pane of the Editor you can:

- ✓ assign a name (Function (=), Location (+), and / or Product (-)) to a component without graphic
  Make sure the "Amount" value is 1, otherwise you generate duplicated components.
  If you want to create a component, spare terminal or cable with a name already existing in the diagrams, the entry will not be created.
- ✓ add an amount to a component without graphic (for example add 1000 labels)
  Make sure not to define a component name in this case, otherwise you generate duplicated components.
- ✓ add new component properties to the components without graphic

4. Select a row on the left pane of the window.
5. In the right pane, open the pull-down list of properties in the last row.



6. Type a text value for the property and press Enter.
   A new row appears automatically, allowing you to add more component properties.

Notes :

*1. You can also add terminals without graphic using this Editor. For further details about creating terminals, see also **Erro! Fonte de referência não encontrada.**.*
*2. You can also add cables without graphic using this Editor. For further details about creating cables, see also **Erro! Fonte de referência não encontrada.**.*

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

To delete a component not in graphic:

1. Select the element.
2. Right-click and select the **Delete Selected** command.
3. Click **OK** to confirm.

Notes:

1. *If you wish to insert these components in the circuit diagram, you can do that through the **Pick List** command in the **Functions** category. Make sure you have defined a database type and a graphic for the element in the channel definition of the type. If in the channel definition no symbol is defined but connection texts and a size for the symbol are set, a black box is generated. If nothing is defined, a rectangle with one connection point is generated when the component is inserted.*

2. *When you position a component without graphic into a diagram, it receives a name in accordance with the IEC 60757 norm, depending on its type (set during its creation) or the type assigned to it from the Type Database. If no specific type is defined (the symbol is simply created as "Component") and no type is assigned from the Type Database, then the name from the Editor is used (the Product (-) field). When you insert a component into a circuit diagram page, it disappears from the Component Without Graphic list.*

*3. If a cable is inserted via the Picklist, you have to follow the rules for drawing cables. If you want to insert cable cores at different places, you can take the first cable core from the Picklist, but after that you have to draw the other cable cores via the **Electrical Cable Cable** command, as the cable disappears from the Component Without Graphic list, as soon as the first core is inserted.*

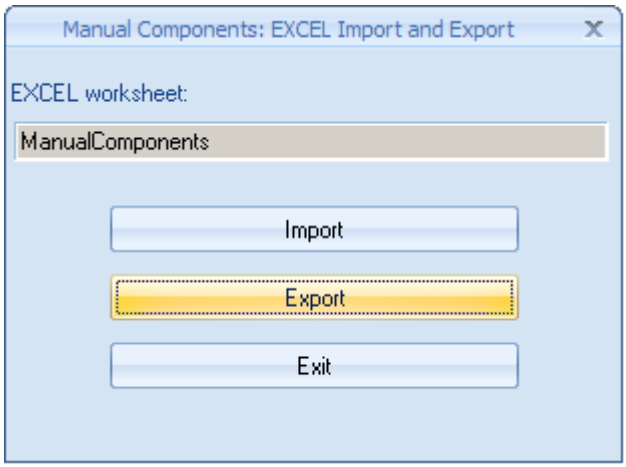
Quick guide: Importing/Exporting manual components from an Excel file

A pop-up command in the Components without Graphic Editor allows you to import/export Excel files, containing information about components without graphics (manual components).

1. Right click in the **Components without Graphic Editor** and activate the **Excel - Import/Export** command.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

The following dialogue appears:



2. Click the **Import** or the **Export** button and select the file to process.
3. Click **Exit** to quit the command.
   Close and reopen the database editor for the modifications to appear.

Hint:

The Excel file used needs a very specific format. We recommend that you create some components without graphic of the desired type and export the list to Excel. Then you can use this file as a master file, put data into it and import it. The first row of the Excel sheet always contains a headline (descriptions of the columns) and can be empty. The second row contains the IDs for identification of the columns. Do not change this row. The third row contains the definitions for the components.

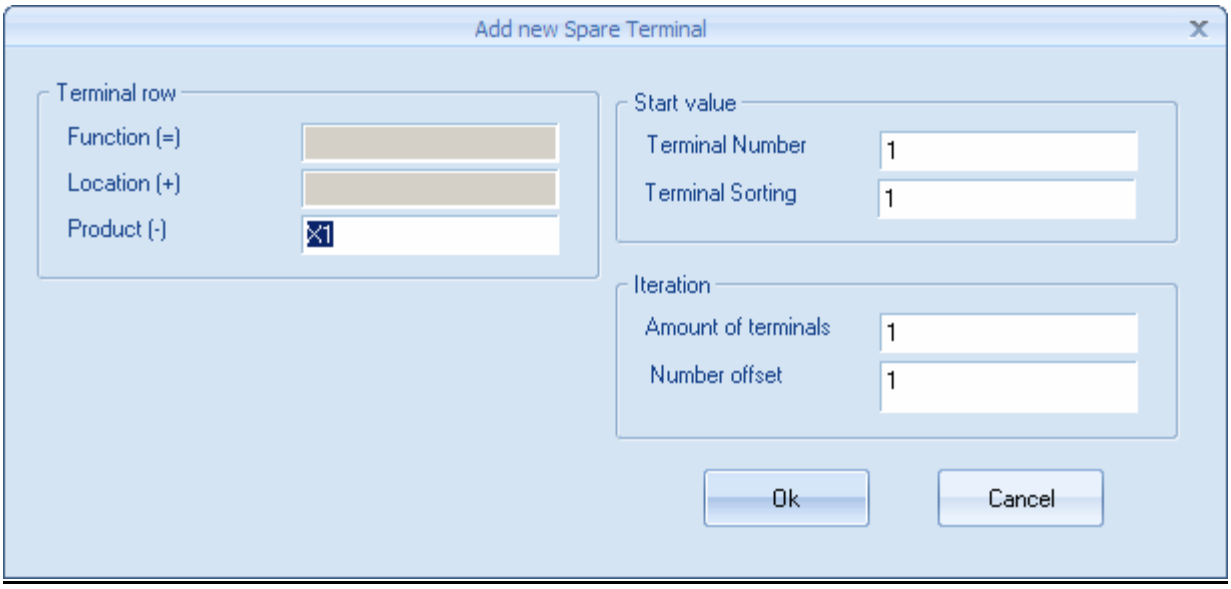
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

OO.2. INSERTING SPARE OR GROUND TERMINALS VIA THE TERMINAL EDITOR

A function allows you to insert spare/ground terminals in the database.

Quick guide "Insert spare terminals":

1. Right click a line in the Terminal Editor list.
2. Select the **Add new component** command.
   The following dialogue appears:



   This window allows you to easily define a lot of spare terminals in a single operation.
3. Add the name of the terminal strip to which the spare terminals will belong. (Function (=), Location (+) and Product (-)).
4. Define the first terminal number and sorting in the "Terminal number" and "Terminal sorting" fields.
   With "Number offset" you define the way, in which terminal number (if numerical value is given) and terminal sorting are to be increased.
   Please keep in mind that a terminal is uniquely specified by a terminal strip, a terminal number and a terminal sorting. If a terminal already exists, it is not generated in here.
   It is not possible to define the single levels of multilevel terminals here.
5. Type in the desired values and click **OK**.
   The spare terminals are inserted in the database list. They also are added to the graphical terminal lists at that position defined by their terminal sorting.

For further details about importing data from an *Excel* file, see also **Erro! Fonte de referência não encontrada.**.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Note:

*You can insert these terminals in the Circuit diagram as well, by using the **Pick List** command from the **Functions** category.*

OO.2.1. HANDLING CABLES WITHOUT GRAPHICS

Quick guide: Inserting cables without graphics:

1. Right click a line in the Cable Editor list.
2. Select the **Add new component** pop-up command.
3. Type in the desired values into the input fields on the right pane.

If you want to add a name for the cable, the name must be unique. If you do not define a name, you can define an amount of cables.
The amount of cable cores is not managed in the case of a cable without graphics. This means that when you insert at least one core in the circuit diagram, the cable disappears from the list of components without graphics. The same happens if you rename an existing cable in the diagram with the name of a cable existing only in the Editor.

Excel import:
When cables are imported from an *Excel* file, and a cable with the same name already exists, the new imported cable receives an extension to its number, separated by "#". For example, if a cable W1 already exists, the "W1" cable imported gets the new name "-W1#001" (if a second cable "W1" is imported, it gets the name "W1#002").

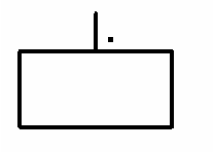
Behaviour in the Pick List:
Let us assume that W1 and W2 are cables without graphics in the *Pick List*.
If you select W1 from the Pick List and change the name to W2, the cable W1 is taken from the Pick List when you select it and the cable W2 is taken from the Pick List, because it is not found in the diagram.
You will never find cables without graphics in the List of Cable-cores, because the cores of these cables do not have targets.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

OO.3. INSERTING SYMBOLS FOR COMPONENTS WITHOUT GRAPHICS IN THE DIAGRAM (PICK LIST)

If you have defined "Components without graphic" (see *Database lists*, *Editor*, *Components without graphic*), you can position the elements via the pick list.

If a symbol for the circuit diagram has been assigned to the type in the channel definition, the given symbol is inserted. If no symbol has been assigned, *SEE Electrical* automatically creates a symbol with one connection point.



When you position a component without graphics into the diagram, it receives the name you have defined in the Editor. If you have not defined a name in the Editor, it gets an automatic name at the insertion. As soon as you position it in the diagram, it disappears from the *Component without graphic List*.

If a cable is inserted via the *Picklist*, you have to follow the rules for drawing cables. If you want to insert cable cores at different places, you can take the first cable core from the Picklist, but after that you have to draw the other cable cores via the **Electrical □ Cable □ Cable** command, as the cable disappears from the *Component Without Graphic list*, as soon as the first core is inserted.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

PP AUTO DIAGRAM

(Advanced)

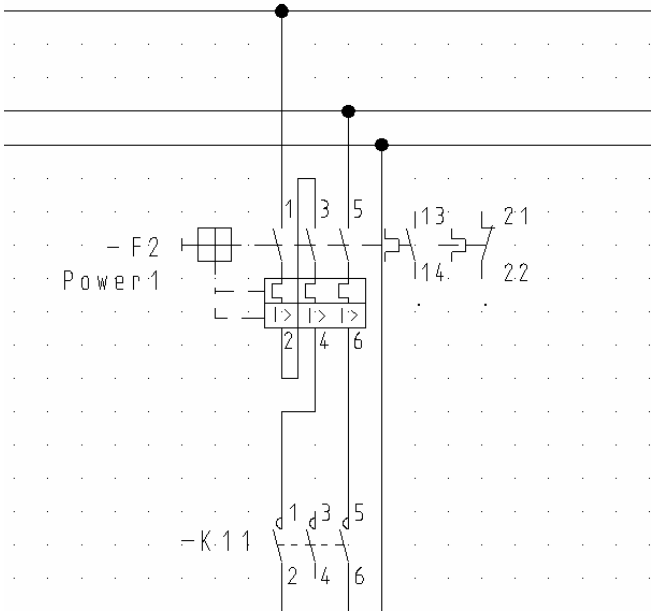
PP.1. INTRODUCTION

The automatic generation of circuit diagrams is executed by using special symbols (groups) and a *Microsoft Excel* spreadsheet.
To use the Autodiagram function, you need to have knowledge about *Microsoft Excel* and about working with *SEE Electrical,* especially about creating symbols and page templates.

PP.2. CREATING SYMBOLS (GROUPS)

You can create symbols easily as they include regular components, their texts and wires and two additional text placeholders to indicate the start-point and the end-point of the group.

1. Create the groups as usual by using components and wires or insert an existing group, ungroup it, if it is a fixed group (if the elements have been grouped using **Edit** **Block, Macro/ Group**).

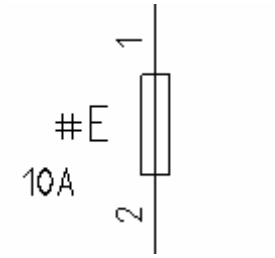


Component name, description and type must be replaced with texts from the Excel spreadsheet by executing the Autodiagram function for the automatic generation of circuit diagrams. Replace the texts at the component with texts from text placeholders.

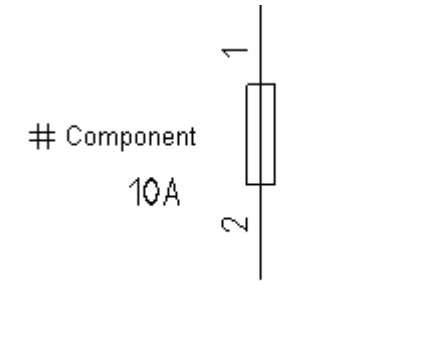
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

You can define placeholder texts in one of the following ways:

2. Specify the column from the *Excel* spreadsheet where the right identification is included (if the column E includes the component name, then a placeholder with format #E is required instead of the component name F2).



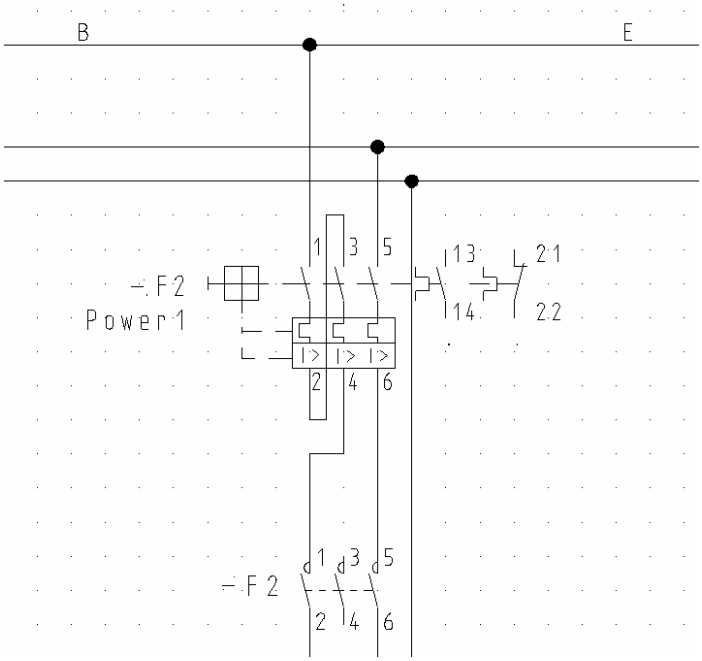
3. Specify an Alias name
Alias names allow entering descriptive names instead of the column names, such as #Component instead of #E. You can assign the names of the *Excel* columns to the alias-names in the Alias sheet within the *Excel* spreadsheet.
By using alias-names, the *Excel* columns can be changed without changing the texts in the components groups. In this case, the Alias-sheet must be adapted.
The use of alias-names facilitates the implementation of text placeholders. However, it is not compulsory to use alias-names.
If you work with alias-names, you have to choose free identification for the component names and to use function/location.



4. Each group needs a marker for the start-point and a marker for the end-point. Over the end marker of the first group, the start marker of the next group is placed, etc. The groups follow each other in a chain. The programme can evaluate whether a group fits into one page. If not, a new page is automatically created.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

5. Use the marker B (Text > Attribute > Other: "**Symbol Start Marker"**) and E (Text □ Attribute □ Other: "Symbol End Marker"). Place appropriate texts (select the **Draw** □ **Elements** □ **New text** command – you will find the text attributes in the "***Attribute***" field under **Other**).

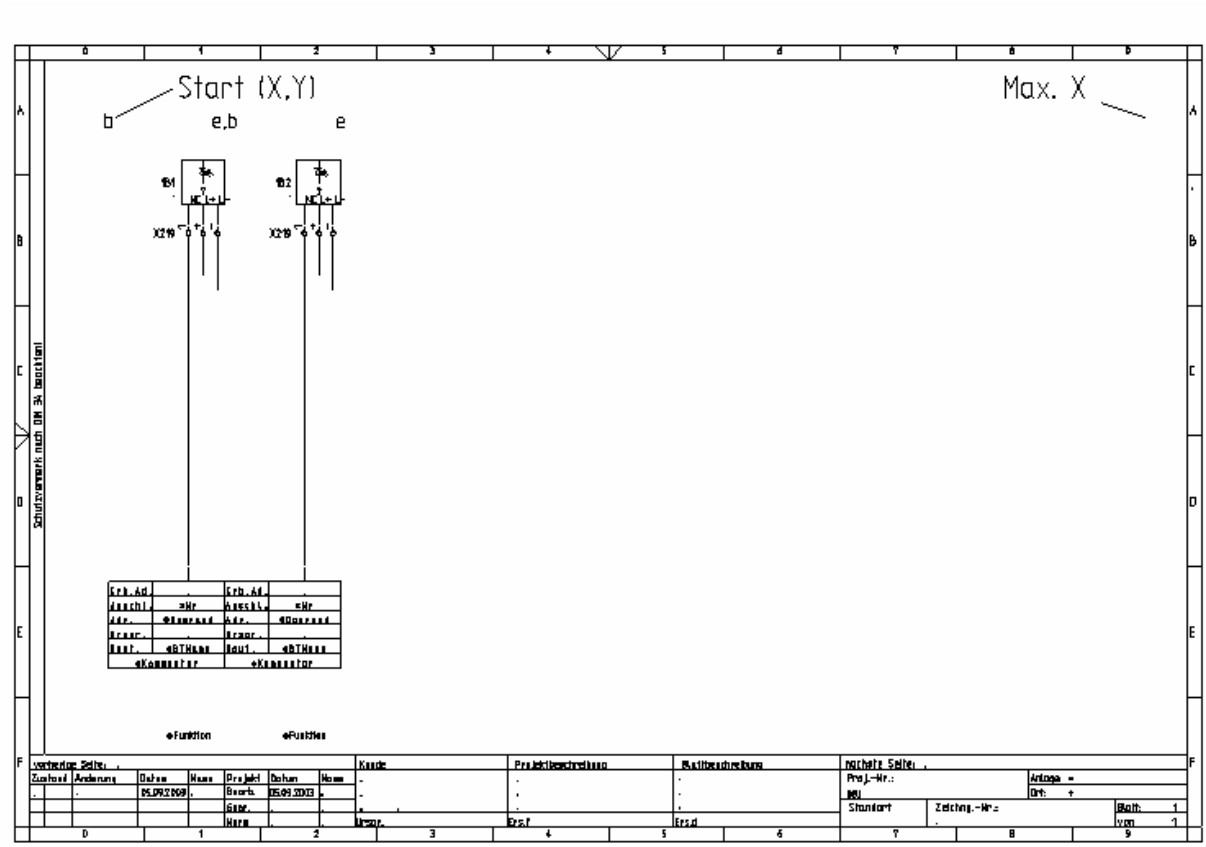


6. Create a group (Select the objects and execute the **Edit** □ **Actions** □ **Block** □ **Block,Macro,Group** command). If the group includes relay coils, do not select the contact mirrors or the contact crosses within the frame.
7. Save the groups in the symbol database which you use only for the automatic generation of Circuit diagrams (select the group or the single objects and drag it into the symbol database – drag the point of the group where the letter B is placed).
All of the groups that could be combined with each other must be saved in the same folder of the symbol database.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

PP.3. PAGE TEMPLATES

You can use any page templates for circuit diagrams but there must be three columns in the template at least.
Through the Autodiagram function, you must define the X and Y position where to place the first group, and the maximal position of X.
The circuit diagram is constructed as follows:



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

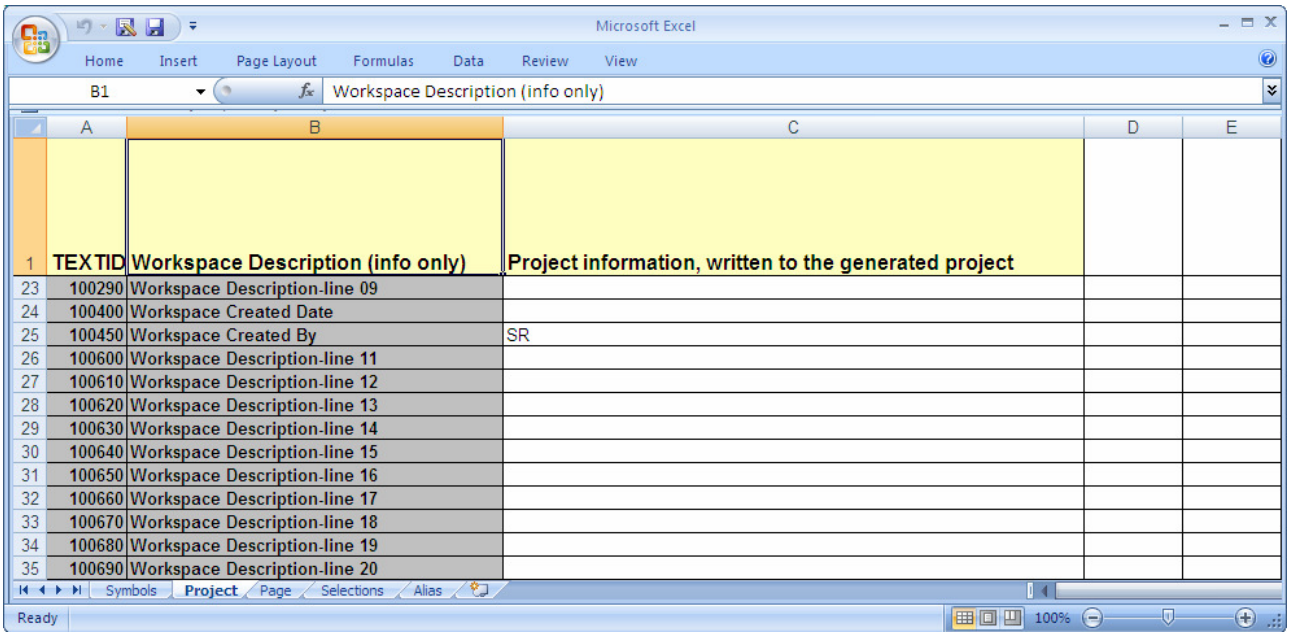
PP.4. EXCEL SPREADSHEET

The *Excel*-spreadsheet includes multiple sheets with different functions. You can change the names of the sheets. The illustration is made by using the included *Excel* spreadsheet *PLC.XLS*.

The *Excel* columns are indicated with letters and the rows – with numbers. Fields are indicated as a combination of the corresponding column name and row name.

PP.4.1. PROJECT DATA (PROJECTINFO)

This area contains information entered in the ***Project information*** window. If the norm sheet contains appropriate text placeholders, then the texts are displayed in the circuit diagrams as well.



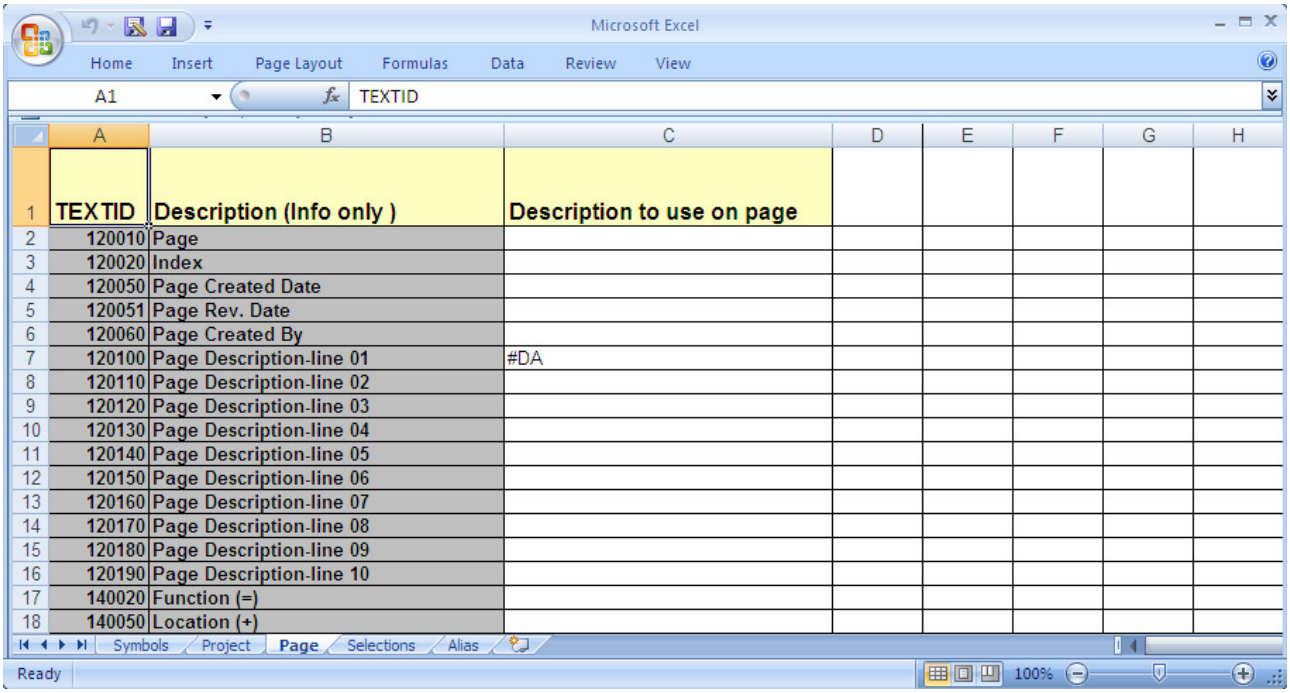
	A	B	C	D	E
1	TEXTID	Workspace Description (info only)	Project information, written to the generated project		
23	100290	Workspace Description-line 09			
24	100400	Workspace Created Date			
25	100450	Workspace Created By	SR		
26	100600	Workspace Description-line 11			
27	100610	Workspace Description-line 12			
28	100620	Workspace Description-line 13			
29	100630	Workspace Description-line 14			
30	100640	Workspace Description-line 15			
31	100650	Workspace Description-line 16			
32	100660	Workspace Description-line 17			
33	100670	Workspace Description-line 18			
34	100680	Workspace Description-line 19			
35	100690	Workspace Description-line 20			

Do not change the "***TEXTID***" column in any case.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

PP.4.2. PAGE DATA (PAGEINFO)

This area contains information entered in the ***Page information*** window for each page. If the applied norm sheet contains appropriate text placeholders, then the texts are displayed in the circuit diagrams, too.



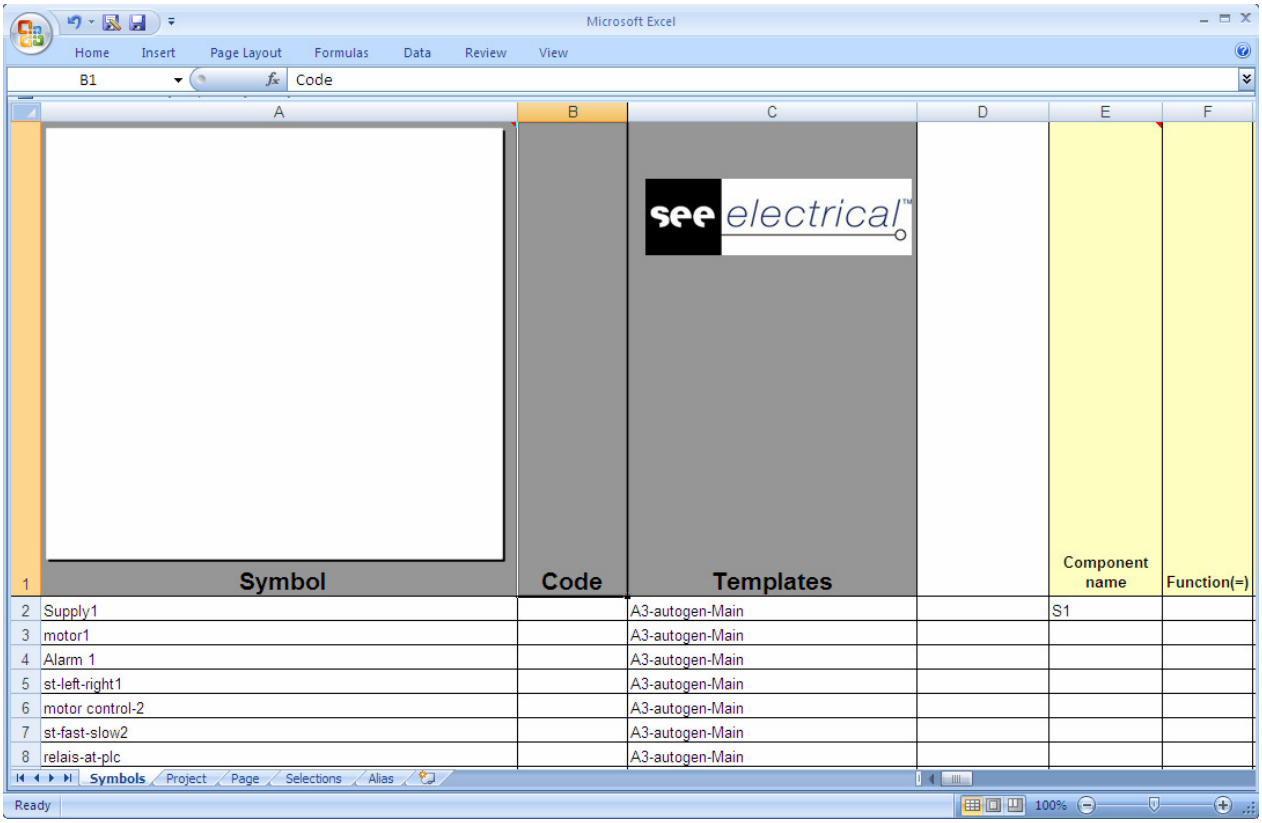
	A	B	C	D	E	F	G	H
1	TEXTID	Description (Info only)	Description to use on page					
2	120010	Page						
3	120020	Index						
4	120050	Page Created Date						
5	120051	Page Rev. Date						
6	120060	Page Created By						
7	120100	Page Description-line 01	#DA					
8	120110	Page Description-line 02						
9	120120	Page Description-line 03						
10	120130	Page Description-line 04						
11	120140	Page Description-line 05						
12	120150	Page Description-line 06						
13	120160	Page Description-line 07						
14	120170	Page Description-line 08						
15	120180	Page Description-line 09						
16	120190	Page Description-line 10						
17	140020	Function (=)						
18	140050	Location (+)						

Do not change the "***TEXTID***" column in any case.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

PP.4.3.SELECT SYMBOL (SYMBOLS)

Specify the groups to be used in the current project.
Specify also whether you want to insert special texts, for example, in the component names and also a description or a type. You can do this if appropriate text placeholders exist in the group (see topic "*Create symbols*" above in this chapter).



Add the groups to be used and delete the unnecessary lines.

Column A (Symbolname):
The column includes the names of the existing symbols. This name must be available in the *SEE electrical* symbol database.

Column B (New page):
If you type the letter "*s*", a new page will be created while inserting the group indicated in this line regardless of whether the new group fits into the current page or not.

Column C (Page template):
Define the page template to use, if a new page is required.
Next columns:
Here, you can define texts for the components. The texts will be inserted by using the automatic generation function if corresponding text placeholders are available in the groups.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

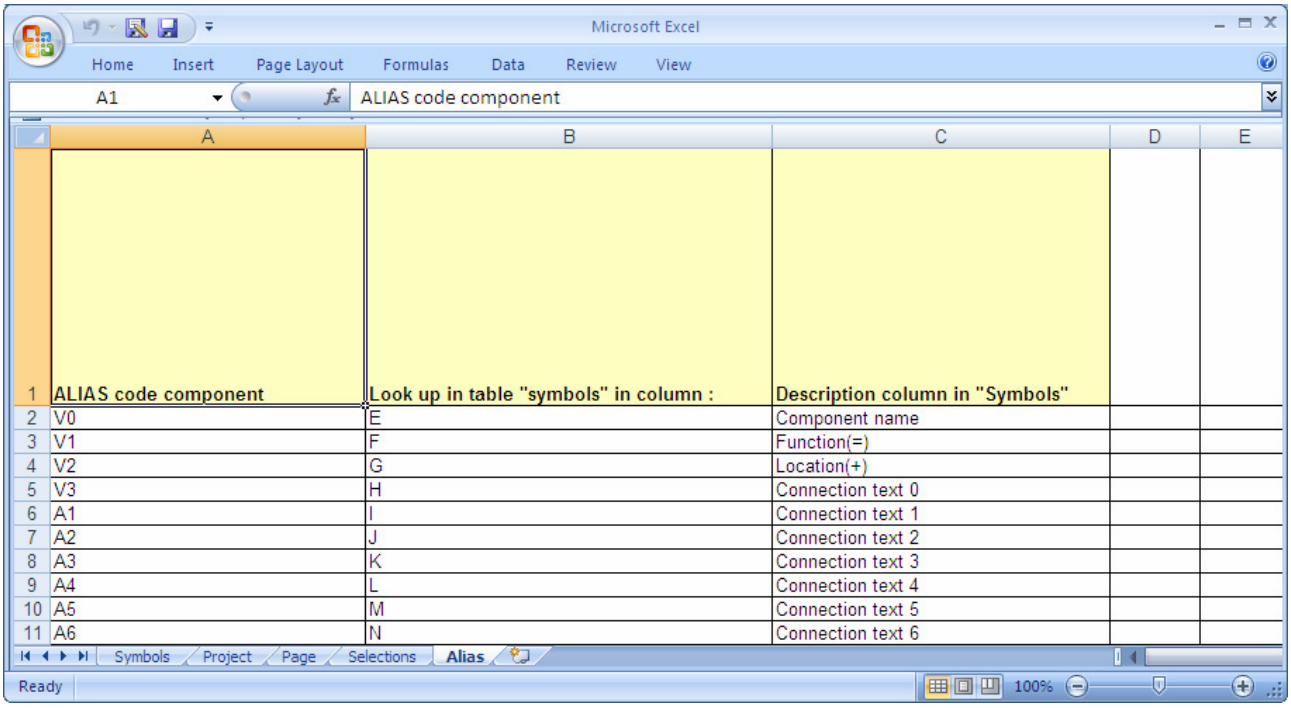
PP.4.4. DEFINING ALIAS NAMES

The groups contain text placeholders, for example for component names etc. that will be filled in with data from the corresponding columns of the *Excel*-sheet Symbols. This text placeholders must be defined as <column identification>, i.e. E, F, etc. You can remember E difficult for the component identification.

Alias names allow entering descriptive texts instead of the column names, for example "*Component Name*" instead of "E", etc. The assignment of the internal column names of the Excel-program to the descriptive alias names is executed in the alias table.

Working with alias names is useful since all of the components can be provided with text placeholders with the Component Name format. It will be easy to change the assignment of the internal column names to the descriptive alias names. If you use text placeholders of the <Column identifier> format in the groups, you can change the assignment only by changing the symbols.

Using alias names facilitates the use of text placeholders, too. Using alias names is not compulsory.



	A	B	C	D	E
1	ALIAS code component	Look up in table "symbols" in column :	Description column in "Symbols"		
2	V0	E	Component name		
3	V1	F	Function(=)		
4	V2	G	Location(+)		
5	V3	H	Connection text 0		
6	A1	I	Connection text 1		
7	A2	J	Connection text 2		
8	A3	K	Connection text 3		
9	A4	L	Connection text 4		
10	A5	M	Connection text 5		
11	A6	N	Connection text 6		

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

PP.5. AUTOMATIC GENERATION OF CIRCUIT DIAGRAMS

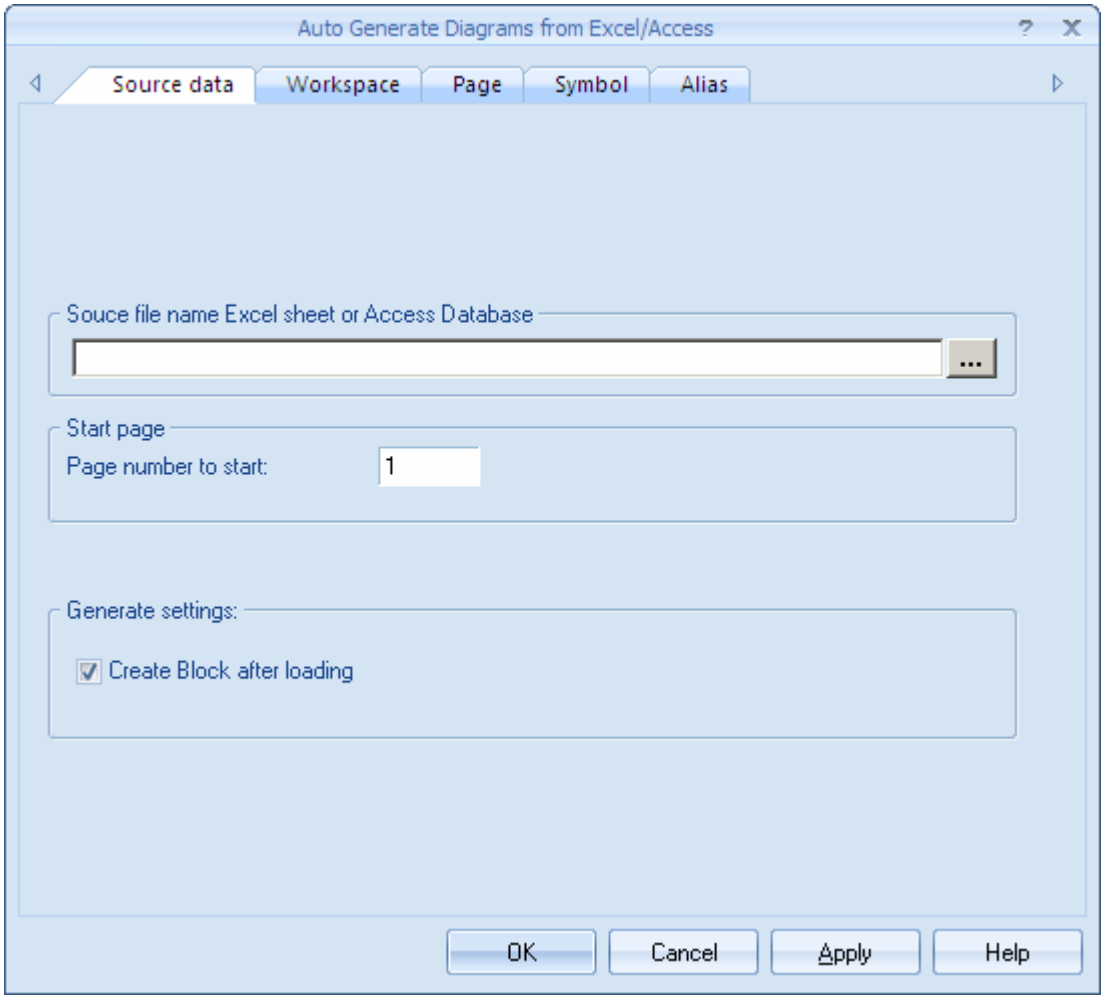
Requirements
Create a new project, for example AUTOMATIC.
If you use an existing project, it will be overwritten, i.e. all existing data will be lost!

Hint:

You can save an empty workspace with a page 1000 as a workspace template.

PP.5.1. THE AUTODIAGRAM COMMAND

If the project is created and the page 1000 is active and visible, launch the **Auto diagram** command. You will find the command in the pop-up menu available for the module "*Circuit Diagrams*".



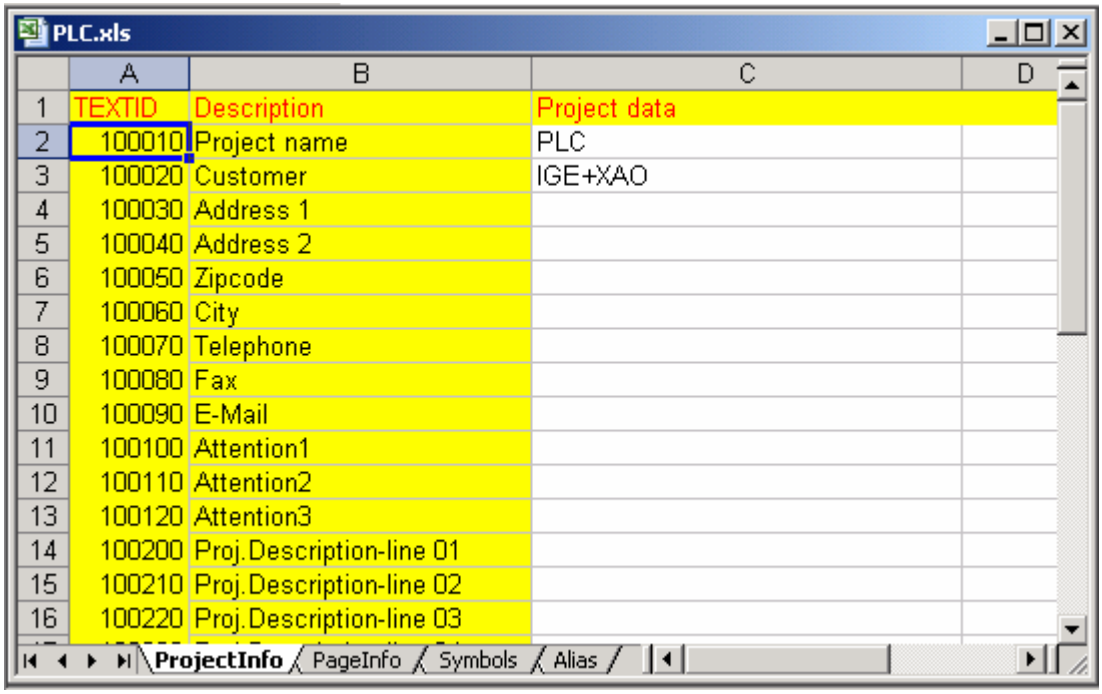
- Enter the required settings (find more details below).
- Click the **OK** button to start the automatic generation of the circuit diagram.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

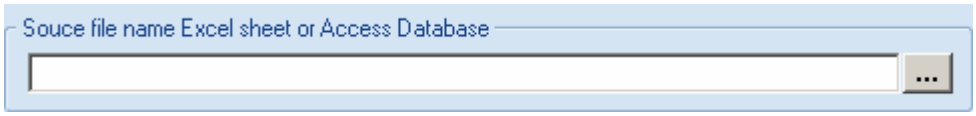
The circuit diagrams will be generated.
The settings are stored within *Windows* and are available for the next execution of the function.

Source data tab

Enter first the sheet of the *Excel*-file and the fields where the data for your project come from.
The included *Excel*-file *PLC.XLS*, sheet **Project data** contains example data:



Source file name (*Excel/Access*)



Enter here the name of the *Excel*- or Access-file, that comprises the defaults about the automatic generation of circuit diagrams.

First page

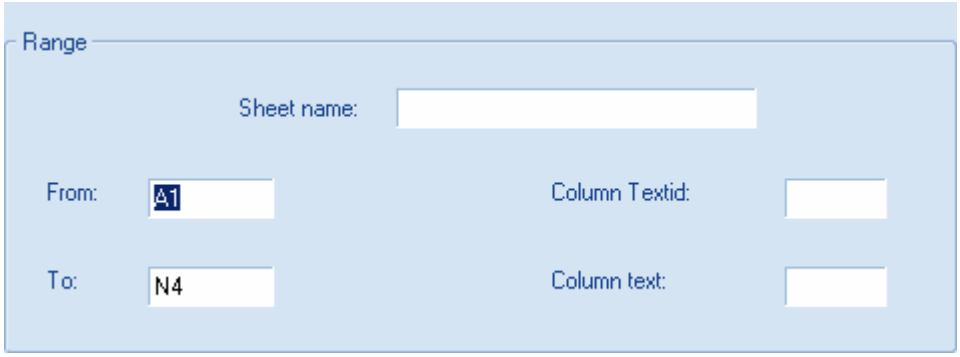
You specify here from which page the automatic project engineering must start.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Attention:

No circuit diagrams must be present after the defined start page number in this field – in such case, they will be deleted.

Workspace information tab



From:
The first field from the Excel-sheet that contains data about the project.

To:
The last field from the Excel-sheet that contains project information.

Sheet name:
For example: *Project data*, name of the Excel-sheet to be used; if you wish to change the sheet names of your Excel-table, you must then type valid names.

Column TextID:
Type the letter of that column of the Excel-sheet containing the TextID data.

Column Text:
Type the letter of the column of the Excel-sheet that contains the entries about the workspace information.

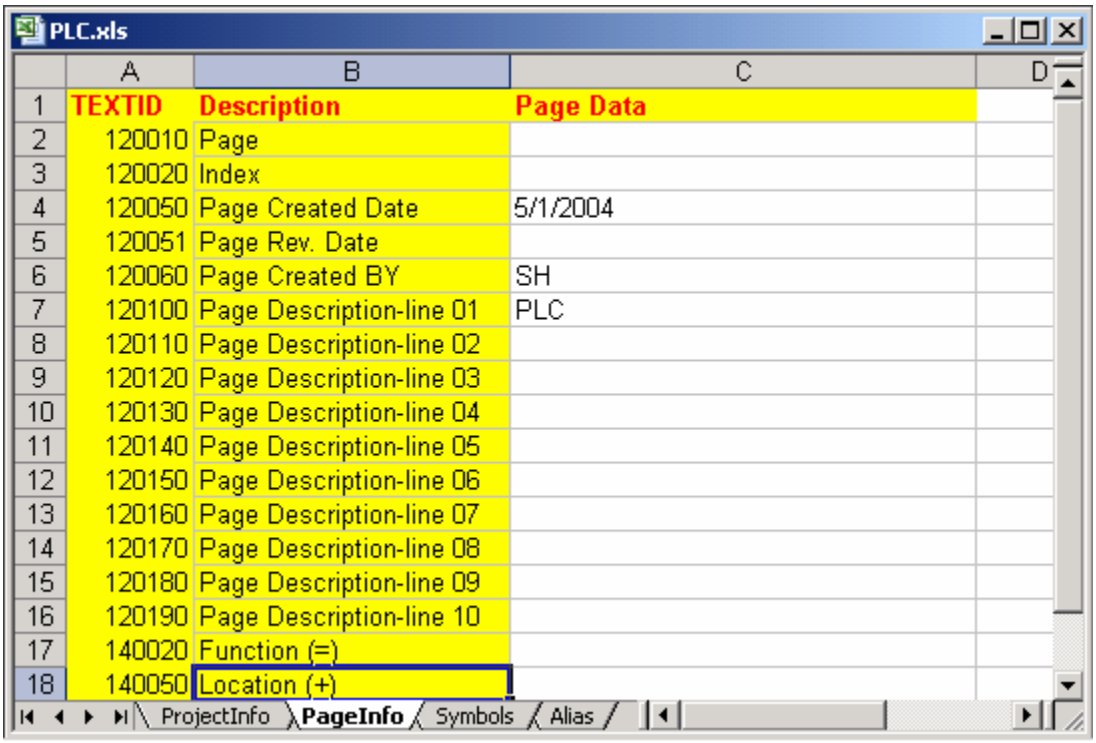
Page information tab

- Enter first the sheet of the *Excel*-file and the fields on this sheet where the page data come from.

All the pages of the project receive the same information.

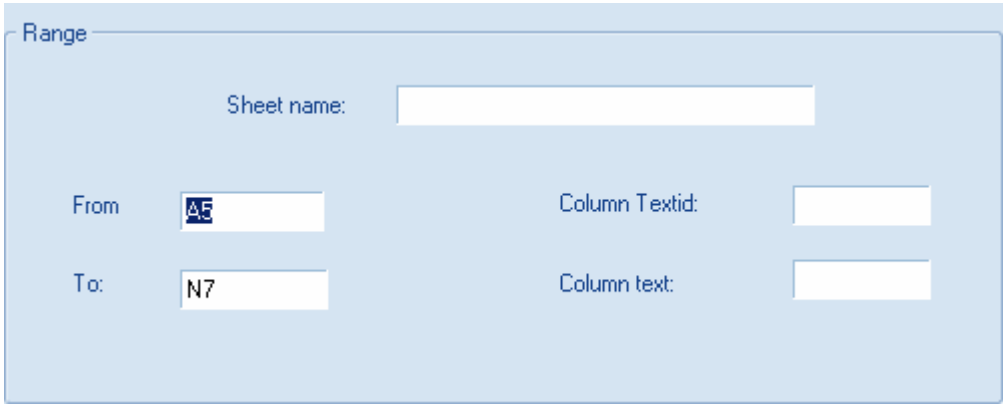
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Example data are illustrated in the Excel-file PLC.XLS, sheet Page Data:



	A	B	C	D
1	TEXTID	Description	Page Data	
2	120010	Page		
3	120020	Index		
4	120050	Page Created Date	5/1/2004	
5	120051	Page Rev. Date		
6	120060	Page Created BY	SH	
7	120100	Page Description-line 01	PLC	
8	120110	Page Description-line 02		
9	120120	Page Description-line 03		
10	120130	Page Description-line 04		
11	120140	Page Description-line 05		
12	120150	Page Description-line 06		
13	120160	Page Description-line 07		
14	120170	Page Description-line 08		
15	120180	Page Description-line 09		
16	120190	Page Description-line 10		
17	140020	Function (=)		
18	140050	Location (+)		

Entries in the ***Page information*** tab:



From:
Type in the first field of the *Excel*-sheet that contains data for page information.

To:
The last field from the *Excel*-sheet that contains data for page information.

Sheet name:
For example: Page Data, name of the *Excel*-sheet; if you wish to change the sheet names in your *Excel*-table, you must type valid names.

Column TextID:
Enter the letter of the column in the *Excel*-table that contains the TextID-records.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Column Text:
Enter the letter of the column in the *Excel*-table that contains the records about Page information.

Symbol data range

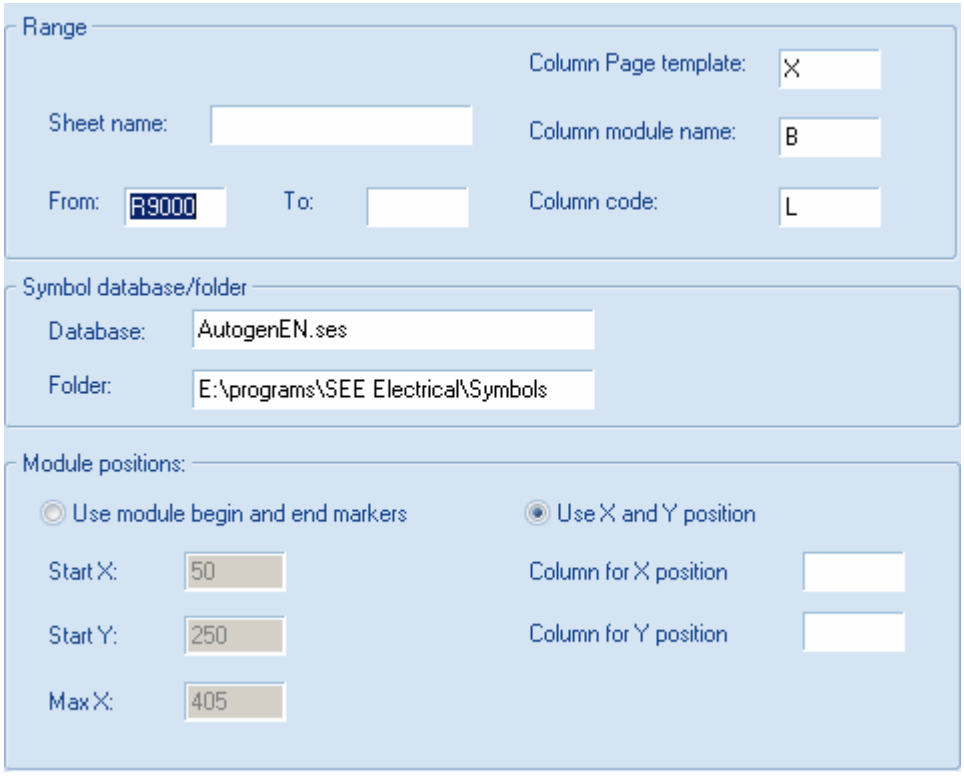
Type the sheet of the *Excel*-file and the fields on this sheet where the data come from about the groups to be placed in.
Input data are illustrated in the *Excel*-file *PLC.XLS*, sheet ***Symbols:***

PLC.xls

	A	B	C	D	E	F	G	H	I
1	Symbolname	New page	Page template		Component	Operand	No	Comment	Description
2	approximation		Electrical Drawing Sheet A3,0-9		112D2	E000.0	2	GS+/EG20.8	Test
3	approximation		Electrical Drawing Sheet A3,0-9		112D2	E000.1	3	GS+/EG20.8	Test
4	blank		Electrical Drawing Sheet A3,0-9		112D2	E000.2	4		Test
5	blank		Electrical Drawing Sheet A3,0-9		112D2	E000.3	5		Test
6	diode		Electrical Drawing Sheet A3,0-9		112D2	E000.4	6	GS+/EG20.11	Test
7	diode		Electrical Drawing Sheet A3,0-9		112D2	E000.5	7	GS+/EG20.12	Test
8	diode		Electrical Drawing Sheet A3,0-9		112D2	E000.6	8	GS+/EG20.13	Test
9	diode		Electrical Drawing Sheet A3,0-9		112D2	E000.7	9	GS+/EG20.14	Test
10	diode		Electrical Drawing Sheet A3,0-9		112D3	E001.1	2	GS+/EG20.15	Test
11	diode		Electrical Drawing Sheet A3,0-9		112D3	E001.2	3	GS+/EG20.16	Test
12	diode		Electrical Drawing Sheet A3,0-9		112D3	E001.2	4	GS+/EG20.17	Test

ProjectInfo | PageInfo | **Symbols** | Alias

Symbol information tab entries:



From:
Type in the first field of the *Excel*-sheet that contains data about symbols.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

To:
The last field in the *Excel*-sheet that contains data about symbols. You could specify a wider range than the just now defined one, in order to avoid changes while adding component groups.

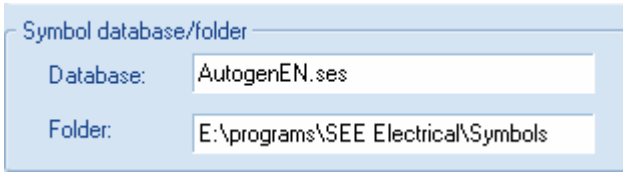
Sheet name:
For example: Symbols, name of the Excel-sheet; if you wish to change the sheet names in your *Excel*-table, you must type valid names here.

Column Page template
Enter here the letter of the column of the *Excel*-table that contains the name of the page template that is to be used if needed.

Column module name:
Determine the letter of the column in the *Excel*-table that contains the symbol names.

Column code:
Enter the column letter in the *Excel*-table that contains "s" for page break.

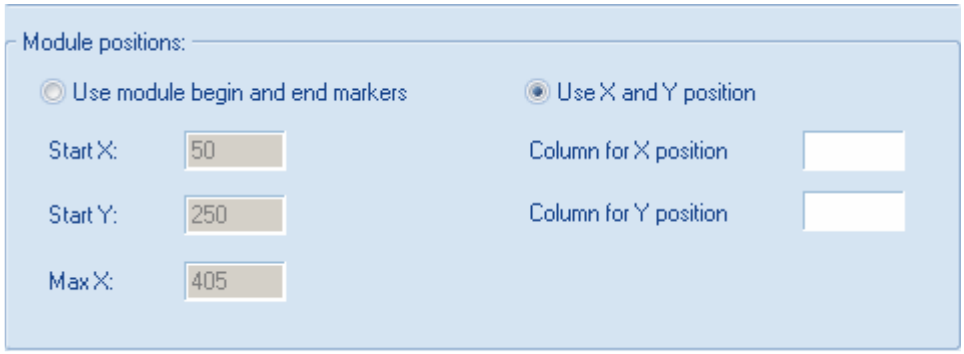
Symbol database/folder area



Database:
Name of the symbol database that contains the groups for the automatic generation of circuit diagrams.

Folder:
Folder in the symbol database that contains all the groups needed for the automatic generation of the circuit diagrams.

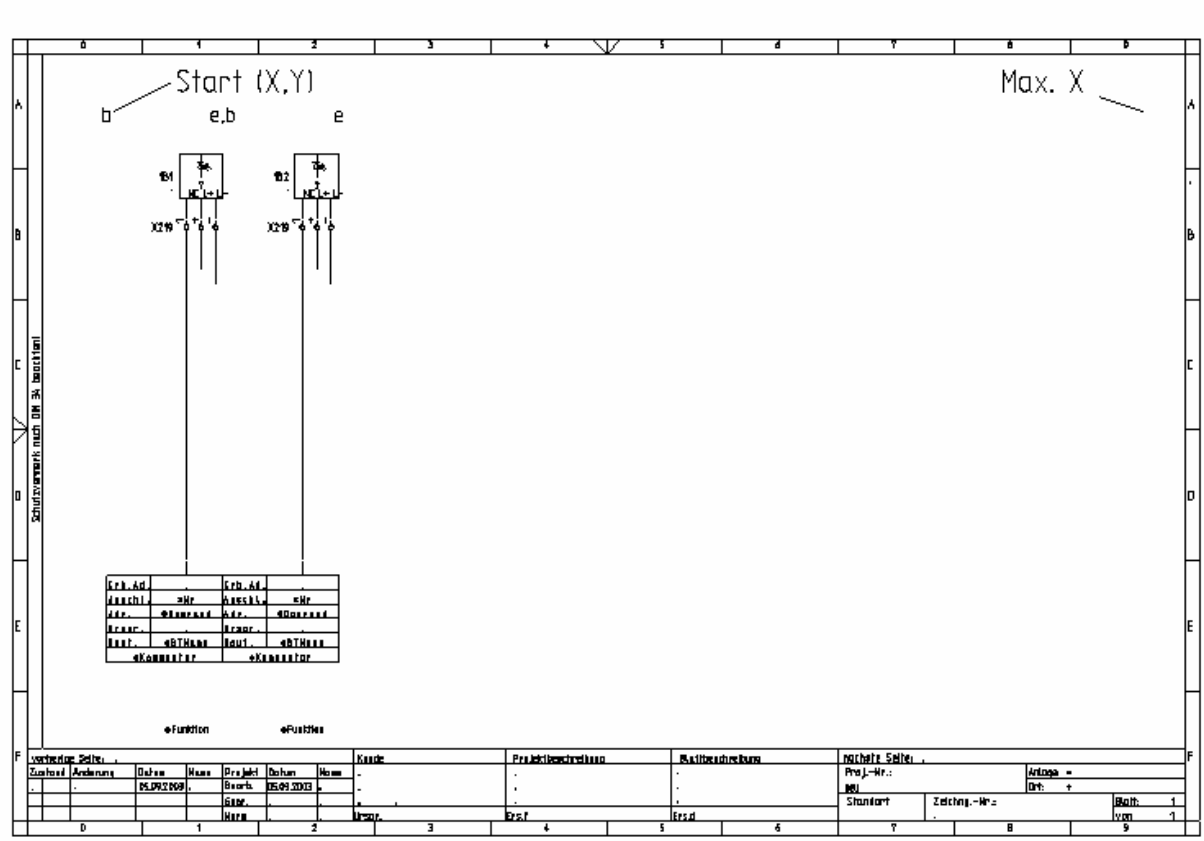
Module positions area



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Activate "***Use module begin and end markers***" to determine the coordinates X and Y where the first group must be placed.

The ***Max.X*** field specifies the maximum area available in the page template. The possibility to place another group in one page depends on your end marker, if it can be placed on a position less than or equal to the value of Max. X. Otherwise, a new page is created.
By selecting the "***Use X and Y position***" option, you can specify the coordinates for the symbol positions from two columns in the *Excel*-list.



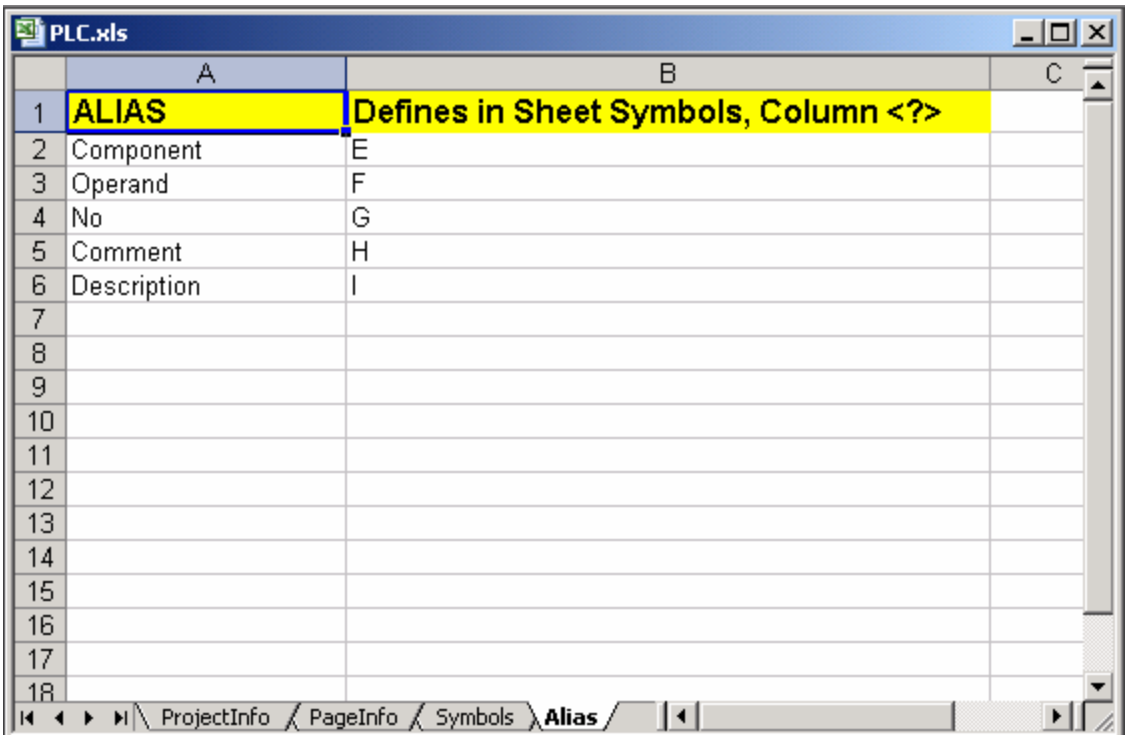
You can define the page template in the *Excel*-spreadsheet. The values in the Module positions area must comport with the used page template.
The number of groups that can be placed in one page depends on the size of the page template and on the size of the groups that you must place, furthermore also, if there is an "s" record in the *Excel*-sheet Symbols that causes a page to break or not.

Alias data range

Use alias names for the text reservation fields, define the sheet of the *Excel*-file and the fields on this sheet where the data must come from.
Using alias names is not compulsory.

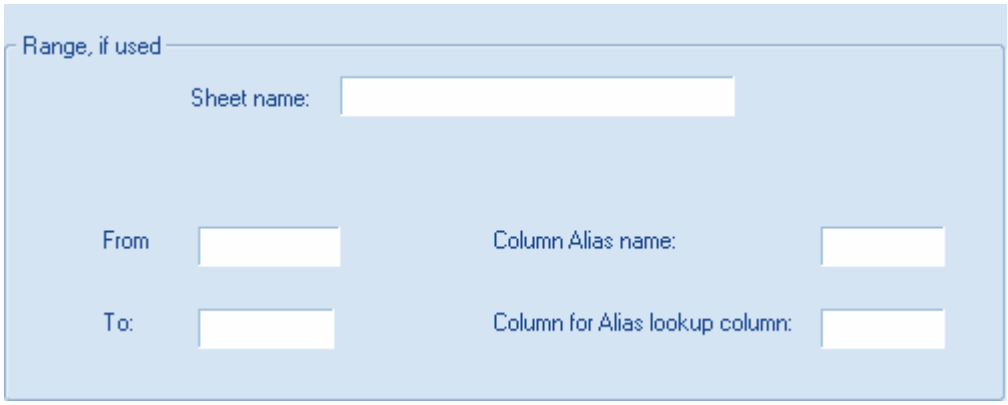
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Data entries are illustrated in the example *Excel*-file *PLC.XLS*, sheet *Alias:*



Alias information tab:

If something is defined here, it is assumed that you use alias names!



From:
The first field of the Excel-sheet that contains relevant information about alias names.

To:
The last field in the *Excel*-sheet that contains relevant information about alias names.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Sheet name
For example: *Alias*, name of the *Excel*-sheet; if you wish to change the sheet names of your *Excel*-table, you must enter valid names here.

Column alias name:
Enter the letter of the column in the *Excel*-table that contains the alias name(for example #Component Name).

Column for Alias lookup column:
Enter here the letter of the column in the *Excel*-table that specifies in which columns of the Excel-spreadsheet the texts for the components must be searched (#T for alias name #Component Name in our example).

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

QQ LISTS AND LABELS EDITOR

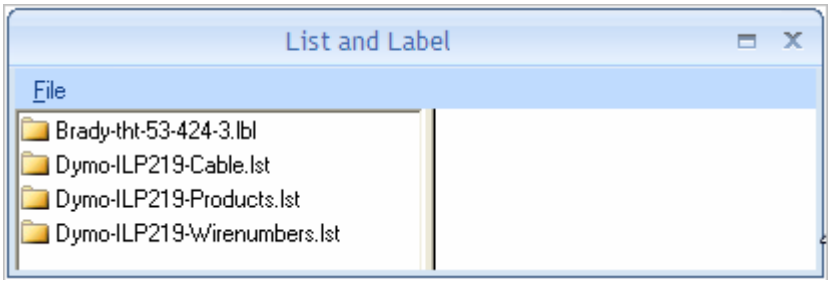
(Advanced)

QQ.1. CREATING LABEL TEMPLATES

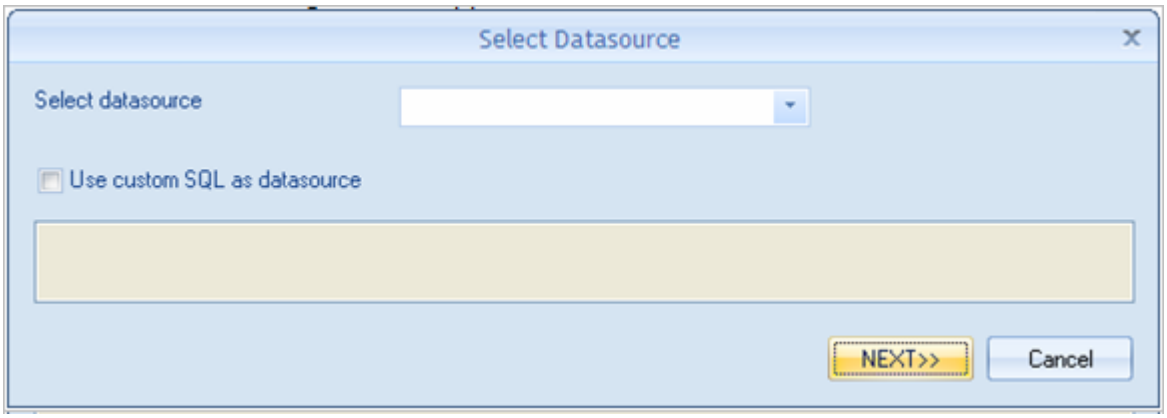
Exercise 39-1: Create a template for labels.

Only one label can be edited. When you print this label, it can be multiplied according to the specified number of lines and columns in the page.

1.CA **File**
2.CO **List and Label**



3.M **File**
Click the **File** menu in the ***List and Label*** window.
4.M **New Label**
5.# <name>
Type the name of the new template.
6.M **Save**
The following window appears:



7.> Select data source
Select the database, from those included in the *SEE Electrical* database lists, which contains the fields you wish to be recorder in the label. Only one list can be selected.
8.> View, Products
9.> Next

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

10.> Next
Click **Next** to close the ***Project Wizard***.

11. Select the printer and other print options.

12.> Next
Close the dialogue box by clicking **Next**.

13.> Select the template for your label that you wish to print.
If the desired template for your label is not in the list, you may select **User-defined** and enter the size of the label manually.
Size of the often used labels can be saved in the CMBTL901.INF file in the folder of *SEE Electrical* (you will find the description of this file structure in the User Manual).
By the "***Only show templates for current page settings***" option you can specify whether all templates have to be displayed or not.

14.> Next
Close the dialogue box by clicking **Next**.

15. The dimensions of the label are displayed.
Now you can make changes as needed. (You have to correct the respective record in the CMBTL901.INF. file). Adjust the format of your label if you have chosen **User-defined**.

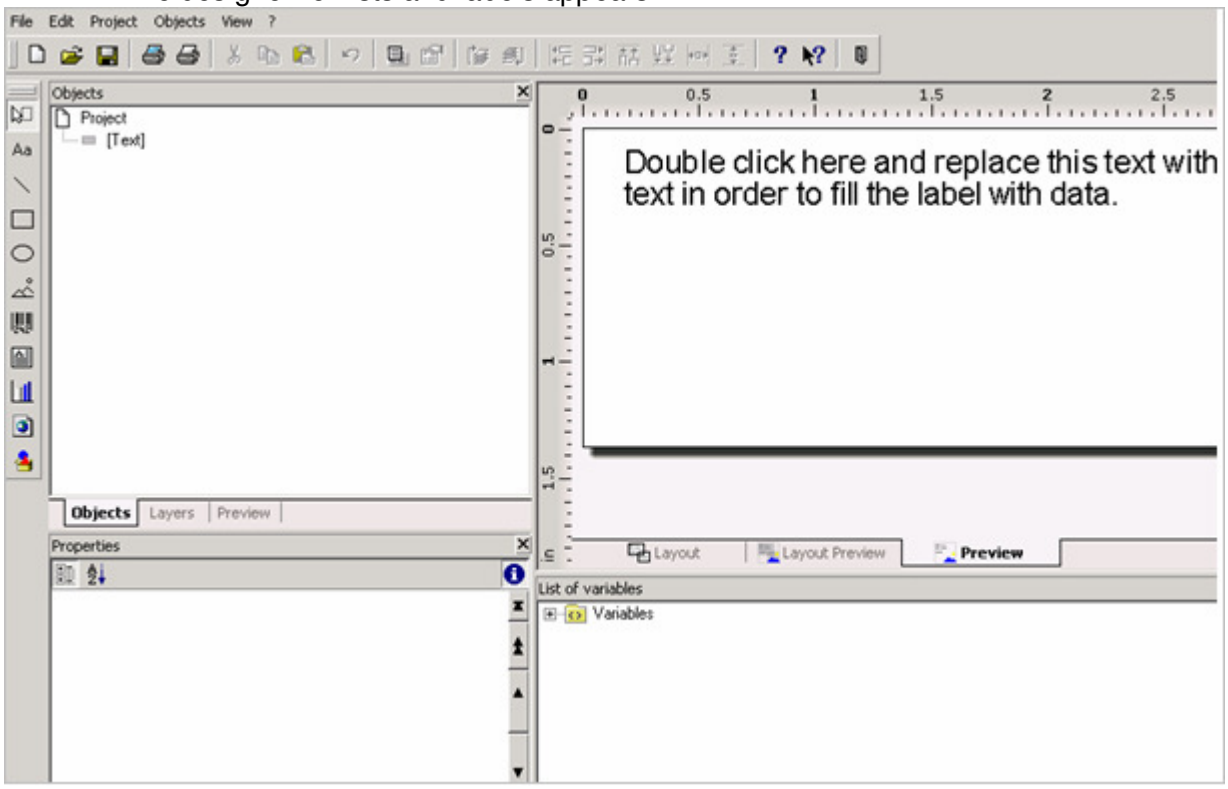
16.> Select the desired Print order.

17.> Next
Close the dialogue box by clicking **Next**.

18.> Next
Close the dialogue box by clicking **Next**.

19.> In most cases, it is not necessary to generate a title line for the labels. Uncheck the "Add title line" option.

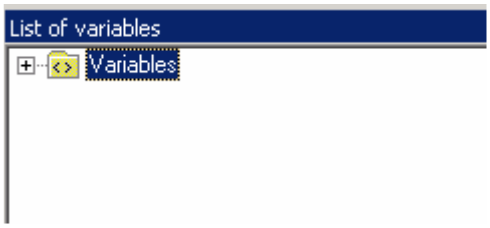
20. Done!
Close the window.
The designer for lists and labels appears:



Select the fields to be printed on your label.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Open the variables list by clicking on the plus sign ⊞:



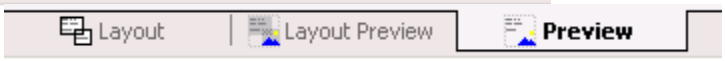
The displayed variables depend on the chosen database list.

21.> Click the desired variable.
You can find information about the list of variables in chapter "*Graphical lists*" in the User Manual where each list is described.

22.+ Drag the variable into the label area above.
You can define the text area using two opposite points as for a rectangle.
You can move this window or change its size. When defining the size of the window, consider the max. length of the text to be printed.
If ***Layout*** is active,

you will see the code number of the text field (for example 160010 for the component name).
If ***Layout Preview***

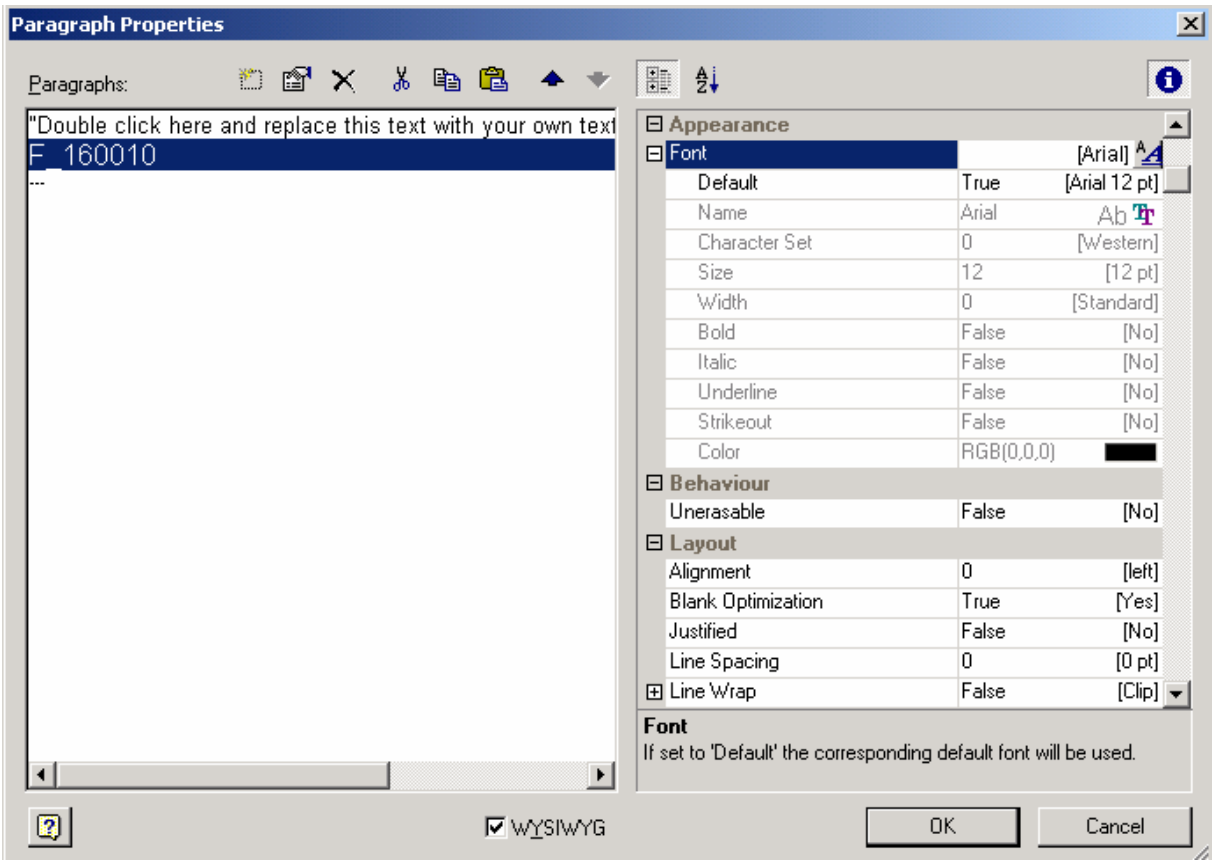
or ***Preview***



is active, you will see the text description of the field, for example "Component name".

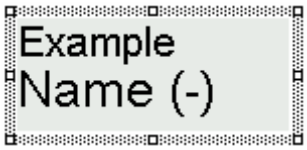
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

23. Double click the text to define attributes and alignment of the text, for example – centred.
The following window appears:



24.> **OK**
Click **OK** to close the window.

25. Choose and place several fields.
Did you place all the fields?
You have another possibility instead of working with blank characters: you can change the size of the available text area via drag points.



26.M **File**
27.M **Save**
28.M **File**
29.M **Exit**
Finish the label template designing.
The template can be used for creating labels.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

QQ.1.1. FILES FOR LABEL TEMPLATES

The label template always contains 5 CDS, LBL, LBP, LBV and ~LBL files. You must back up these files. If you want to create a new label template using an existing template, copy these 5 files in *Windows Explorer*, a "Copy of .... *" is created.
Then, you can rename the label template in the start window of the ***List and Labels Designer***.

QQ.1.2. CHANGING LABEL TEMPLATES

If changes of the *Label Layout* are required after the label has been created (for example, the result after printing does not match the label or it is not positioned correctly), you can change your label template.

- Open the List and Label Designer, click your label and select Design from the pop-up menu.
- Click Next to reconfirm the chosen database. The List and *Label Designer* appears.
- Choose **Project** □ **Page setup** and then, in the ***Layout*** window, select the ***Page setup*** tab to change the position and the size of the label. Click the ***Printer Selection*** tab to change the printer, too.

QQ.1.3. PRINTING LABELS

- Open the start window of the *List and Label Designer* and double-click the template.

Please, make sure that the print options are set correctly.

- Select **Start** position to specify where to start printing (line 2, label 3 left). This setting allows continuing after a paper break. Click **Select**, and then click on the field to start.

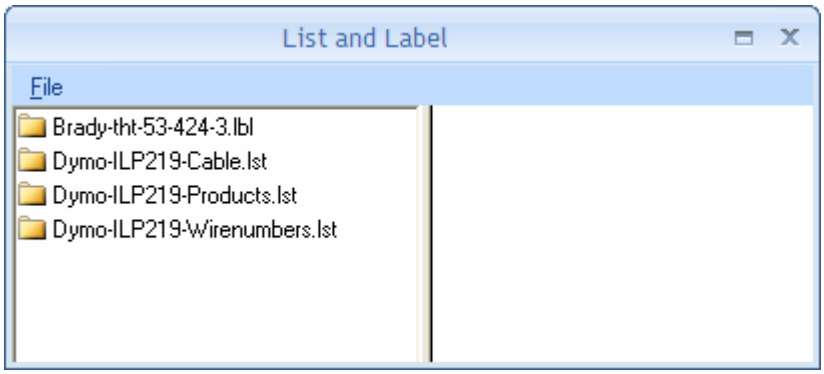
Then, print the labels.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

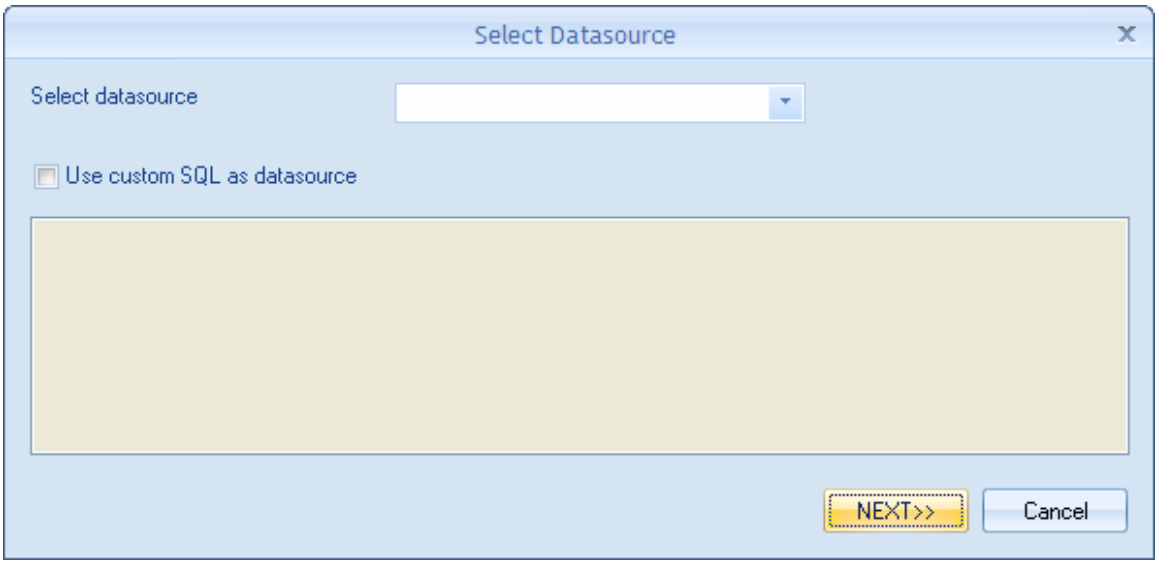
QQ.2. CREATING LIST TEMPLATES

Exercise 39-2: Create a list template.

1.CA **File**
2.CO **List and Label**



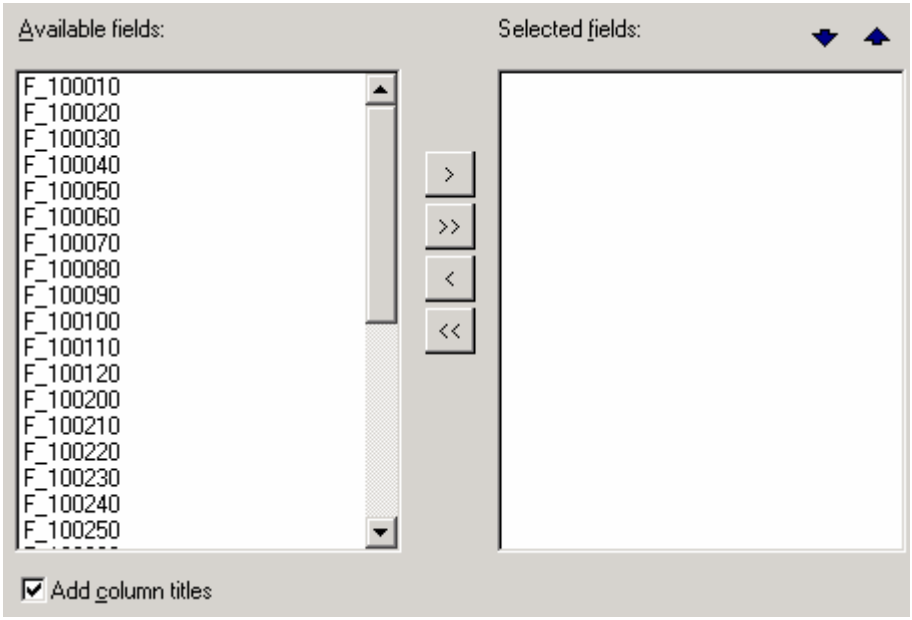
3.M **File**
Select the **File** menu in the ***List and Label*** window.
4.M **New List**
5.# <name>
Type the name of the new template.
6.M **Save**
The following window appears:



7.> Select data source
Select the database, from those included in the *SEE Electrical* database lists, which contains the fields you wish to be recorder in the label. You can select only one list.
8.> Products
9.> Next
10.> Next
Press **Next** to close the ***Project Wizard*** window.

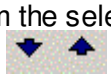
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

11. You can specify whether to use one printer for all pages or to use different printers for first page/following pages.
12.> Next
After you have made your choice, click **Next** to close the dialogue box.
13.> Specify the printer and the other printing options.
14.> Next
Close the dialogue box by clicking **Next**.
15. You can define page numbering and add title.
If you want to add a title, specify if it is to be printed on the first page or on all pages.
16.> Next
Close the dialogue box by clicking **Next**.
17.> Select Zebra pattern for the table to print the lines between rows and columns. You can choose the Create summary option. The summary displays the number of records only on the last page or on all the pages. Specify whether to create summary on all the pages or on the last page only.
18.> Next
Close the dialogue box by clicking **Next**.
19.> Choose the fields for the list:



(The fields in the list depend on the chosen database list, example: Product). You will also find common fields in the list, such as project name, project editor etc. You do not have to select these common fields but place them in the header line.
Choose to **Add column titles** or not before you select the fields.

Click then the desired fields and click on the icon to move them into the **Selected fields** area.

You can remove fields from the selection by clicking on the left arrow icon . You can select multiple fields and move them to the selection by clicking on the right arrow icon . The icon moves all the fields to the selection and the icon removes all the fields from the selection.
Using the  arrows, you can arrange the selected fields as desired.

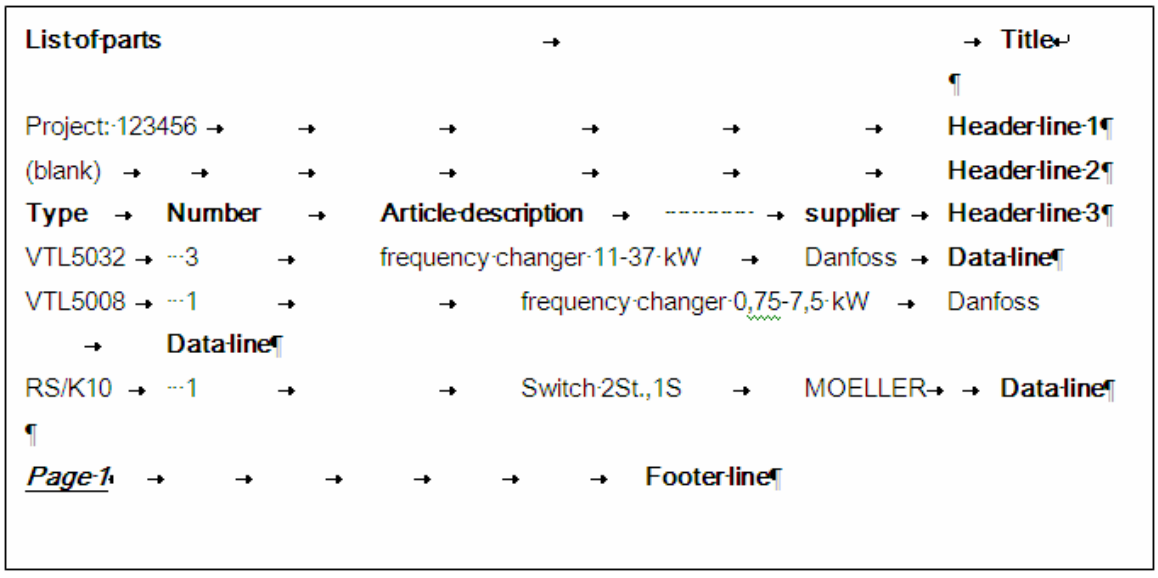
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

You can find the variable list in the chapter **Graphical lists** of the **User Manual,** where each list is described.
If the fields have been selected in the desired order, click "Done!" to close the dialogue box.
The list template is created.
In the next steps, you will find hints about changing the created list.
Did you place all the fields?

20.M **File**
21.M **Save**
22.M **File**
23.M **Exit**
Finish creating the template.
The template can be used for generating lists.

List templates include :

- ✓ *List title (text object)*
- ✓ *Header lines (in tables)*
- ✓ *Group Header (in tables)*
- ✓ *Data lines (in tables)*
- ✓ *Group Footer lines (in tables)*
- ✓ *Footer lines (in tables)*
- ✓ *List Footer lines (in tables)*



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Example of a list without groups:

List of parts			**Title**
Project: 123456			**Header line 1**
(blank)			**Header line 2**
Supplier: Danfoss			**Group Header 1**
Type	**Number**	**Article description**	**Group Header 2**
VTL5032	3	frequency changer 11-37 kW	**Data line**
VTL5008	1	frequency changer 0,75-7,5 kW	**Data line**
Supplier: MOELLER			**Group Header 1**
Type	**Number**	**Article description**	**Group Header 2**
RS/K10	1	Switch 2St.,1S	**Data line**
Page 1			**Title**

QQ.2.1. FILES FOR LIST TEMPLATES

The list template contains 5 CDS, LST, LSP, LSP and ~LST files. You must back up these files. If you would like to create a new list template by using an available one, copy the 5 files using the Explorer; a "copy of ...". *" is created.
Then, you can rename the list template in the start window of the *List and Label Designer*.

QQ.2.2. LIST TITLE (TEXT OBJECT)

List titles are created automatically if you have activated the "***Add title***" option in the *Project Wizard*.
The list titles can consist of more than one line.
The list titles cannot contain parts of *SEE Electrical* projects (such as project name, project created date, editor, etc.)

QQ.2.3. HEADER LINES

Header lines in a table are used to define column titles, if you are not working with a group structure.
Header lines can contain parts of *SEE Electrical* projects (such as project name, project created date, editor, etc.).

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

QQ.2.4. GROUP HEADER

Groups are used for structuring the list, for example, if all the products of a manufacturer must be printed consecutively, and the name of the manufacturer has to appear as a group header.
You must define a condition for the group change in the header, i.e. if data in the specified field(s) change, a new group starts. You can choose whether to continue on a new page and show the group header again or not.
The group header may consist of more than one line, for example, a title and column titles.

QQ.2.5. GROUP FOOTER LINES

If you structure the list using groups, the group footer line can contain the article total price or the article total length. Besides, it can contain only texts.

QQ.2.6. DATA LINES

Data lines contain fields that must be entered in the list.
The available fields depend on the chosen database list of *SEE Electrical*. However, the available fields vary in the database lists too. For example, the "Free Text 01" field appears in the list of products only if at least one component in the current project contains a text with such attribute.
If a desired field is not available for a data line, you must choose another database list or a combination of two database lists (see below the definition of SQL-queries), or you must use a project that contains the needed fields.

QQ.2.7. FOOTER LINE

If you do not work with groups, the footer line can include the total price of all the products or their total length. Besides, it can contain only texts.

QQ.2.8. LIST FOOTER LINE (TEXT OBJECT)

Footer lines are generated automatically if you have activated the page numbering option in the *Project Wizard*. Footer lines can contain more than one line.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

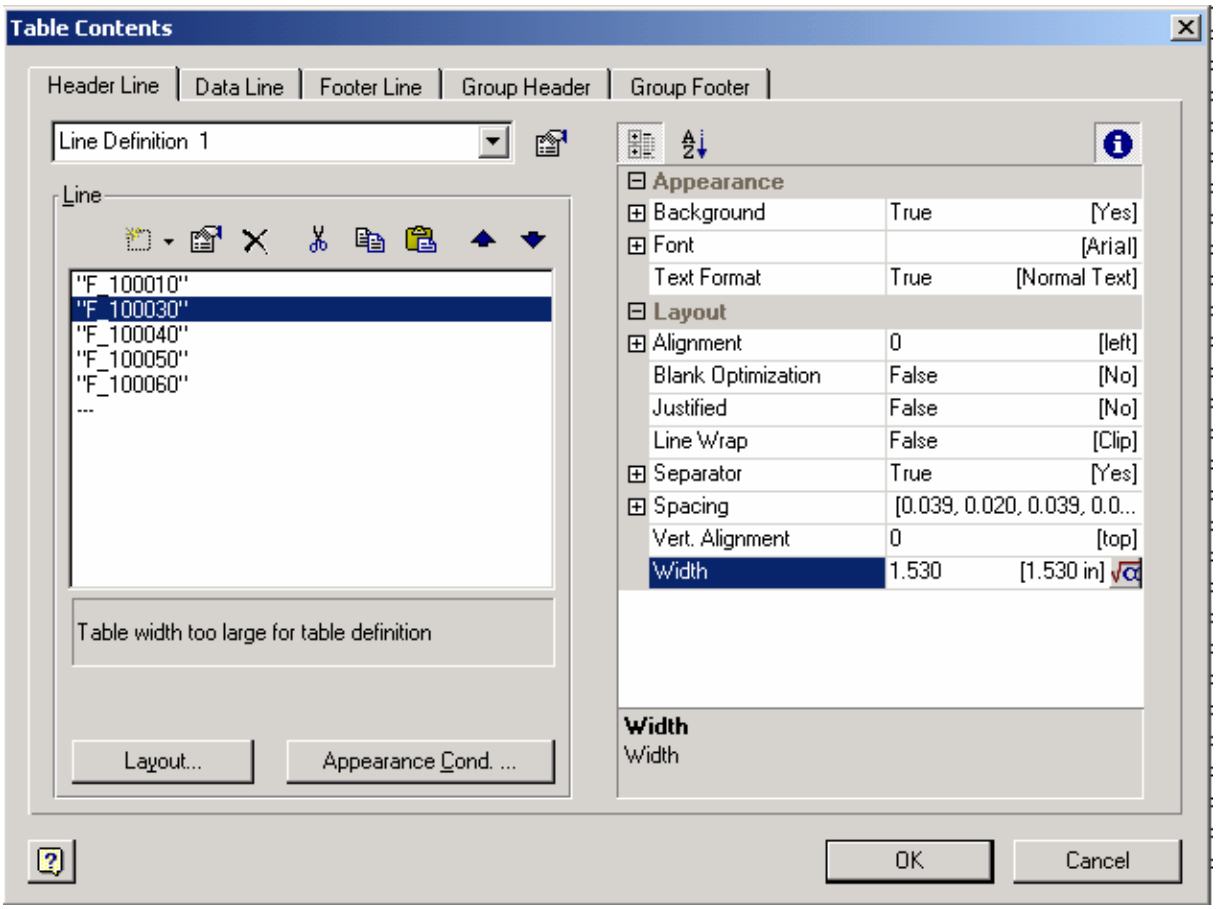
QQ.3. EDITING LISTS

Exercise 39-3: Change the available template.

1.CA **File**
2.CO **List and Label**
3.> Select the template to change
4. Right-click with the mouse. The pop-up menu appears.
5.CO **Design**
6.> Next
Changes are to be made in the ***Layout Preview*** area, as you see here, not only the field number as in the ***Layout*** area, but also the text content of the field. You can activate the ***Layout Preview*** tab in the bottom window's border.

QQ.3.1. CHANGING COLUMN TITLES (HEADER LINES)

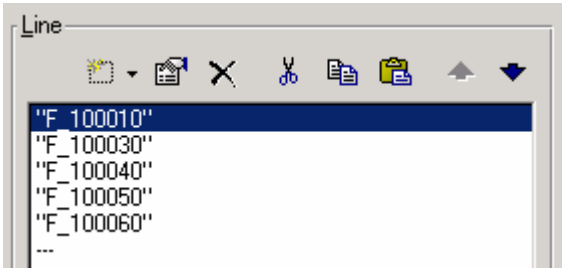
1. If you edit a list template, you can change column titles by double-clicking it.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

2. In the ***Table contents*** window □ ***Header Line*** tab window, you can change the Layout of the text or the Font in the right window area, i.e. the size and the name of the font.

   To change the column title, double-click on it, for example "F_100010", if you want to enter a header line, for example "Project name".



3. If you double-click any field, the following new window appears:

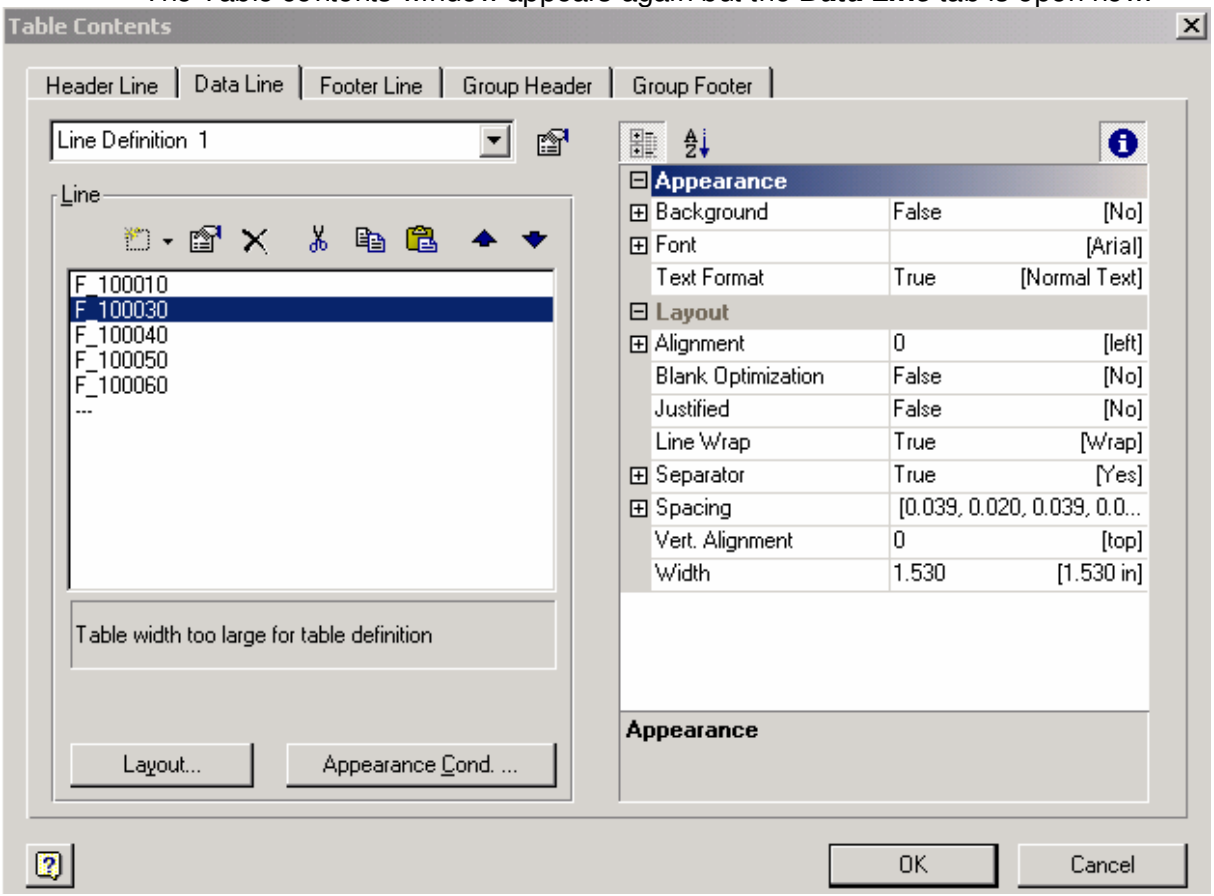
4. Double-click again and type the desired text. The text must be enclosed within double quotation marks ".

5. Close the dialogue box by clicking **OK**.
6. Change all header lines of the columns in this way and then click **OK** to close the ***Table Contents*** window.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

QQ.3.2. CHANGING A LIST (FONT SIZE, FIELDS ORDER, ADDING OR DELETING FIELDS)

1. If the list template is open, double-click one of the fields in the table.
The Table contents window appears again but the ***Data Line*** tab is open now.



Click to change the layout of a table line (Print margins, the font preference of the text). Besides, you can change the ***Font*** and the ***Layout*** in the right window's area.

Click the or arrows to move line down or to move line up.

After a field selection, click the icon to delete the selected field. After you have closed this window by clicking **OK**, you must double-click the column header line of the table and delete the corresponding record.

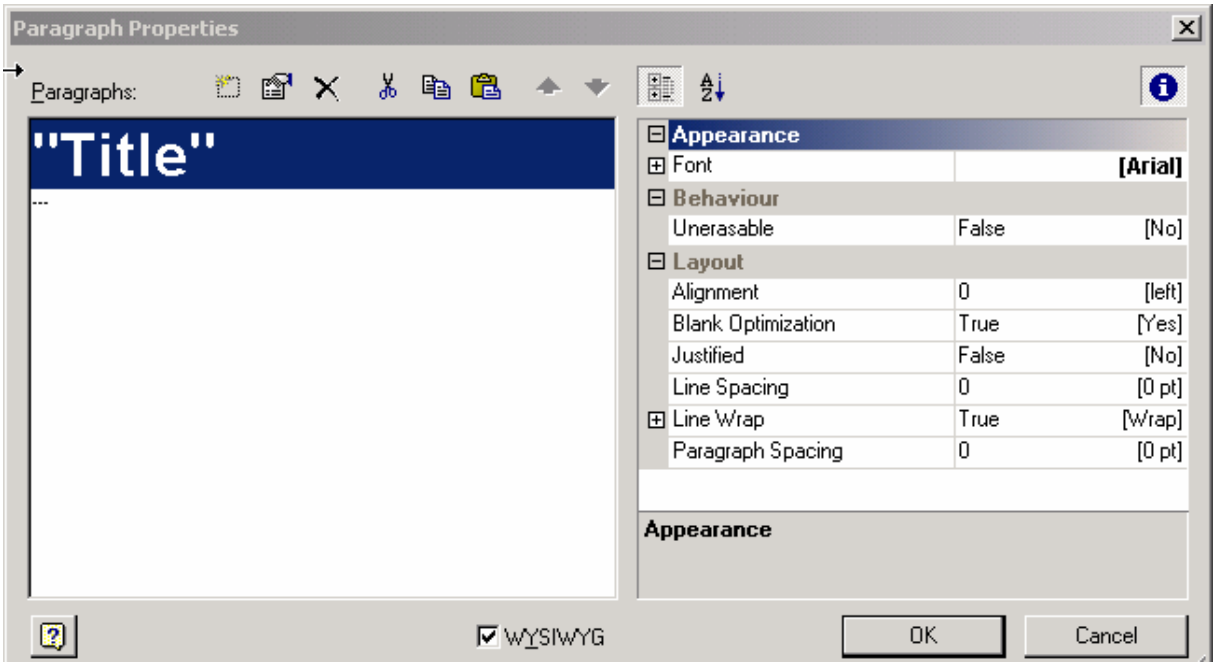
Click on the icon to insert a field. The ***Edit Table*** window appears. You can double-click the *Fields* folder in the upper left pane and double-click the desired field code to select it.

Click **OK** to close the ***Edit Table*** window**.** You must afterwards double-click the column header line in the table, insert the corresponding record and put it in the right place.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

QQ.3.3. CHANGING THE LIST TITLE

1. You can edit the list title by double-clicking on it.
2. The following window appears:



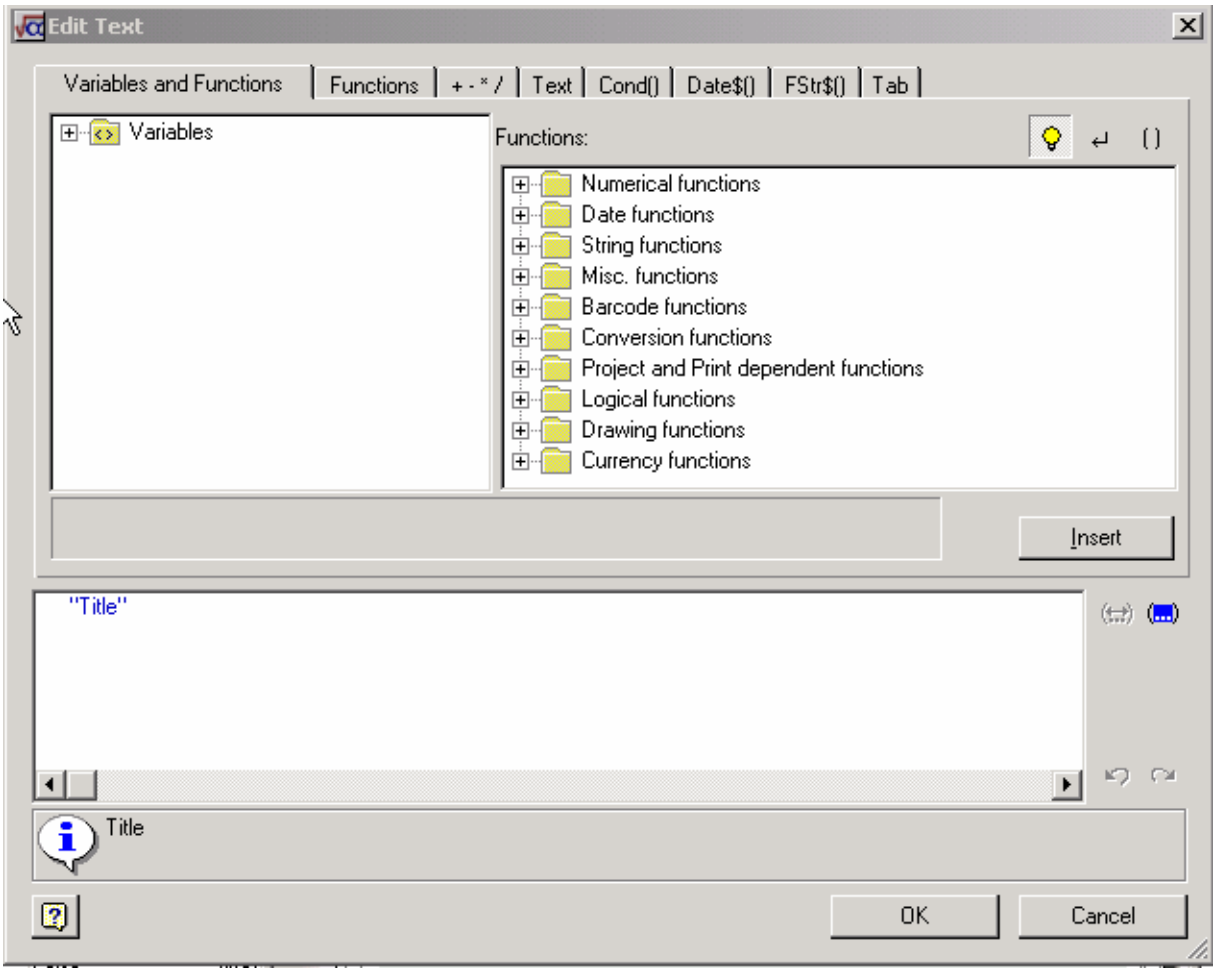
You can change the alignment of the list to appear centred, make the font **Bold** or Underline, change the type and size of the font (see properties under ***Font*** or ***Layout*** in the right area of the window).

You can delete the list title by clicking on the icon.

You can insert an additional list title by clicking on the .

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

4. If you wish to change the text in the list title, double-click the text again –in the above example, double-click Title. The following window appears:



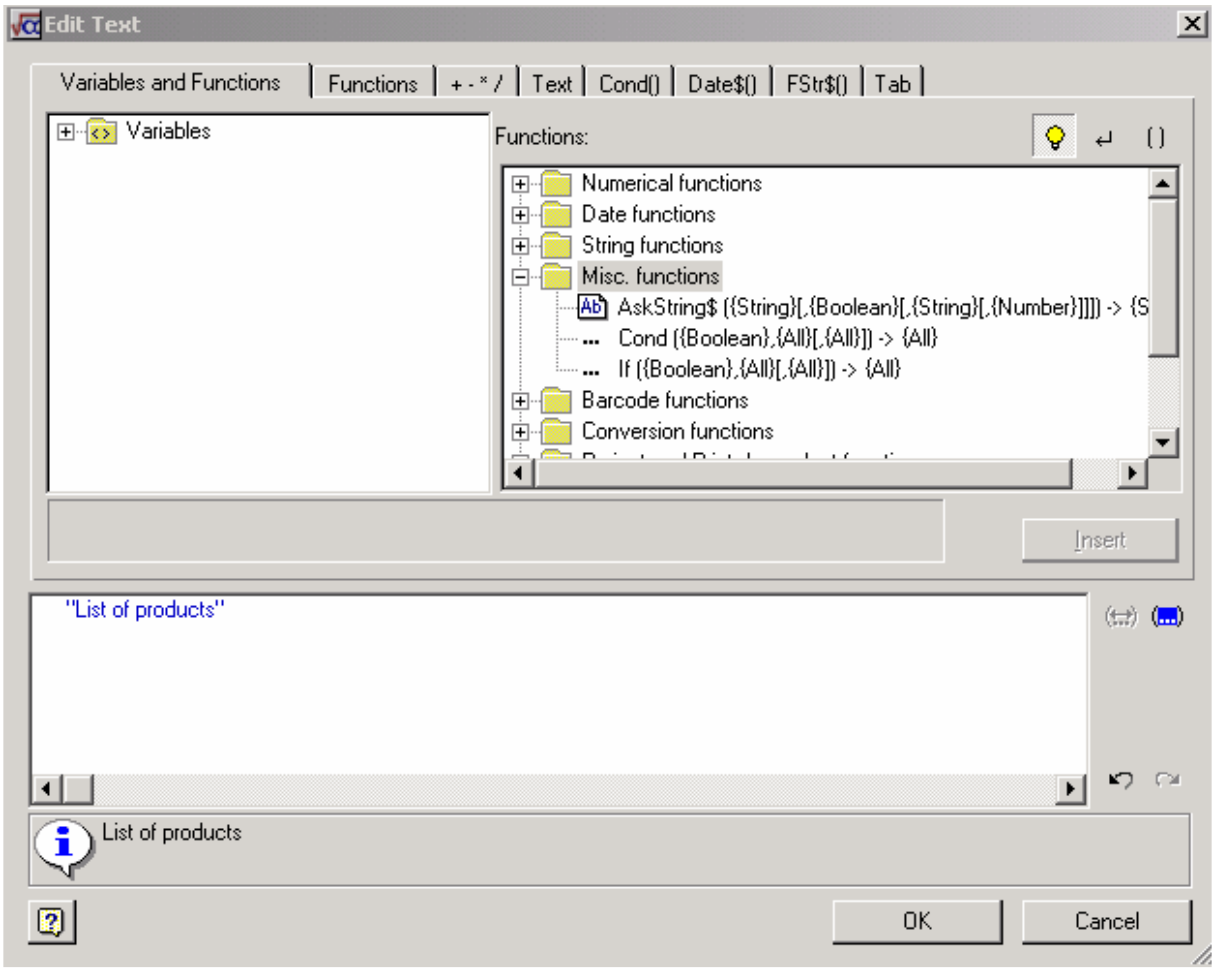
Change the text by double-clicking on it again. The texts must be enclosed within double quotation marks, for example "List of products".

3. Click **OK** to close the window.
4. Close the ***Paragraph Properties*** window by clicking **OK**.

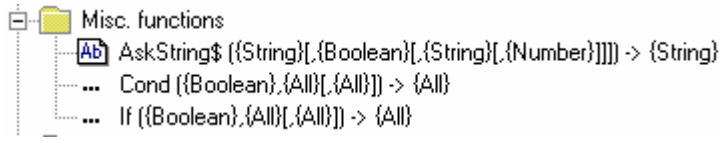
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

QQ.3.4. LIST TITLE WITH QUESTIONS

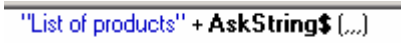
1. If the window shown in the following illustration is open, you can define questions to appear while creating the list. (If the window shown in the following illustration is not open, double-click the list title and the Paragraph Properties window will appear. Double-click the list title again.)



2. Define a question about the project name.
Click behind the text that must be displayed in the header line, e.g. "*List of products*".
Open then the *Miscellaneous functions* folder in the right window pane by double-clicking it.
Select "AskString$ {...} again by double-clicking it.



3. The variable is displayed behind the text.

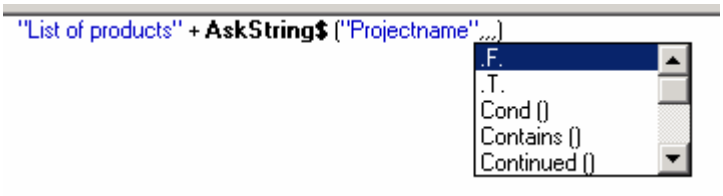


The variable "AskString {...} allows you to create a question by entering user-defined texts.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

You must enter the arguments, too.
Move the cursor to the place between the next two characters of the variable: Type then how the variable must be named, for example, you can type "project name". The text must be enclosed within double quotation marks ".

Move the cursor forward behind ("Projectname",



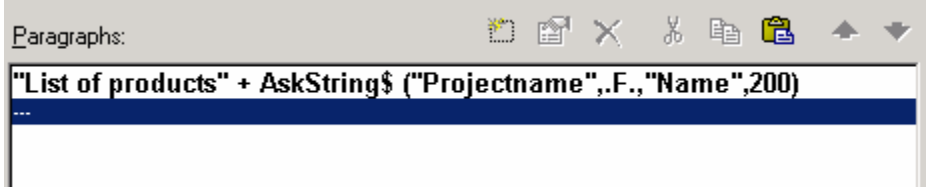
Choose ".F." from the automatically displayed pop-up menu to specify that the dialogue with the question will appear only once, at the beginning of the list.
Move the cursor forward to the position ("Projectname",.F., . Enter the value to be suggested automatically, for example "Name", as the entry of the project name is expected. The text must be within double quotation marks ".
Give the max. character number for the text. We suppose 200 characters for the project name. Move the cursor to the position between the characters,).

Now, the variable for the question about project name (or another user-defined text) before generation of the list has been defined.

4. Click **OK** to finish editing the first header line.

QQ.3.5. LIST TITLE WITH GENERATION DATE

- Enter the date of generation in the title of the list.
- Double-click the title.
- Double-click onto … beneath the line of the list title "Products + AskString$("Projectname",.F.,"Name",200).



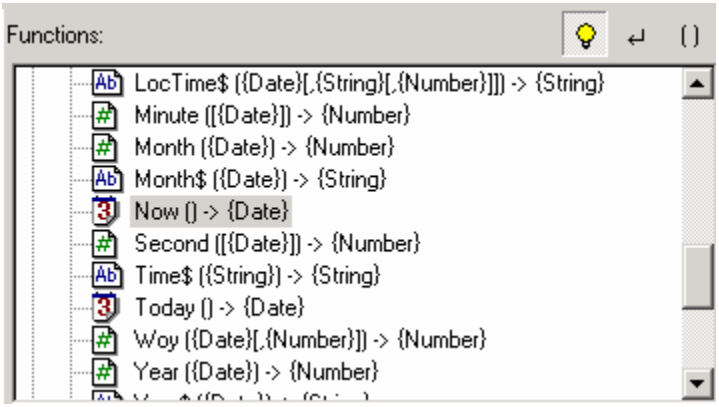
You will go into the window again where you can edit the texts. For example, type the text "created:" on the keyboard in the text area.

- Click then behind the text (position the cursor behind the text).
- Open the "*Date functions*" folder in the "***Functions***" area by clicking the plus sign.
- Scroll the suggested variables until you see "Now() ->{Date}".

This variable shows the current day.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

- Double-click on it.



The function is transferred to the text line.

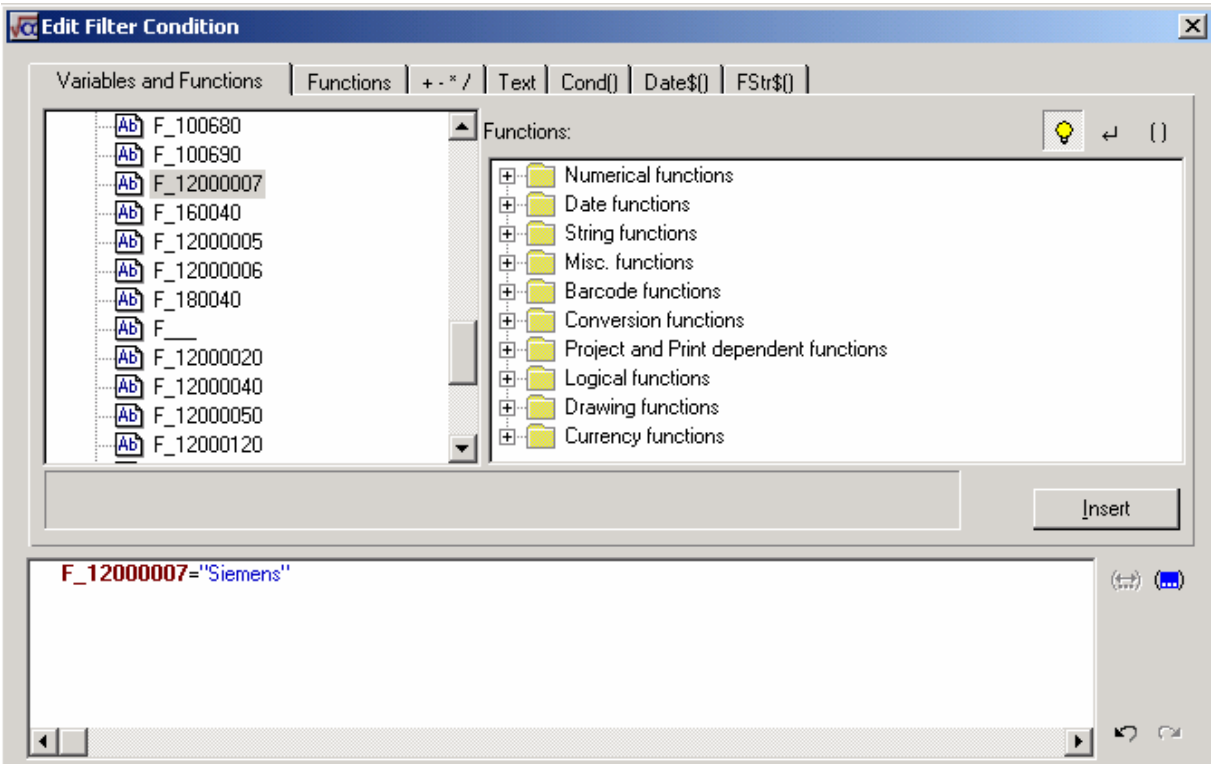
QQ.3.6. FILTERING IN THE LISTS

It is possible to define multiple filters. By means of some examples, the definition of frequently used criteria will be explained.

QQ.3.6.a. LIST OF PARTS FOR A PARTICULAR MANUFACTURER

1. If the fields of the list template have been defined (List of parts), define then the filter using the **Project** □ **Filter**.
2. Double-click the *Fields* folder in the left window's area to open it and double-click the field code you like to define a filter for. In the example, this is the manufacturer and the code is F_12000007.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



The field code is displayed in the bottom window's pane.

3. Type here the manufacturer's name for the filter, behind a "=" sign. The name must be between double quotation marks ". Uppercase and lowercase are taken into consideration.

You can type the expression "Ask String ...", too (see below), for example.
F_12000007 = AskString ("Manufacturer,.F.,"SIEMENS",200)

The "Ask String*$* ..." executes a question about the manufacturer.
You will find an illustration about the "Ask String$ ..." expression in the chapter "*List title with questions*" above how to enter the project name manually in the list.
If you wish to be sure that the manufacturer will be found after using uppercase or lowercase, you can use the command "Upper$...":
F_12000007 = Upper$(AskString$("manufacturer,.F.,"SIEMENS",200))
The expression "Upper$..." converts all the letters into uppercase.

4. Close the dialogue box by clicking OK and save the list template.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

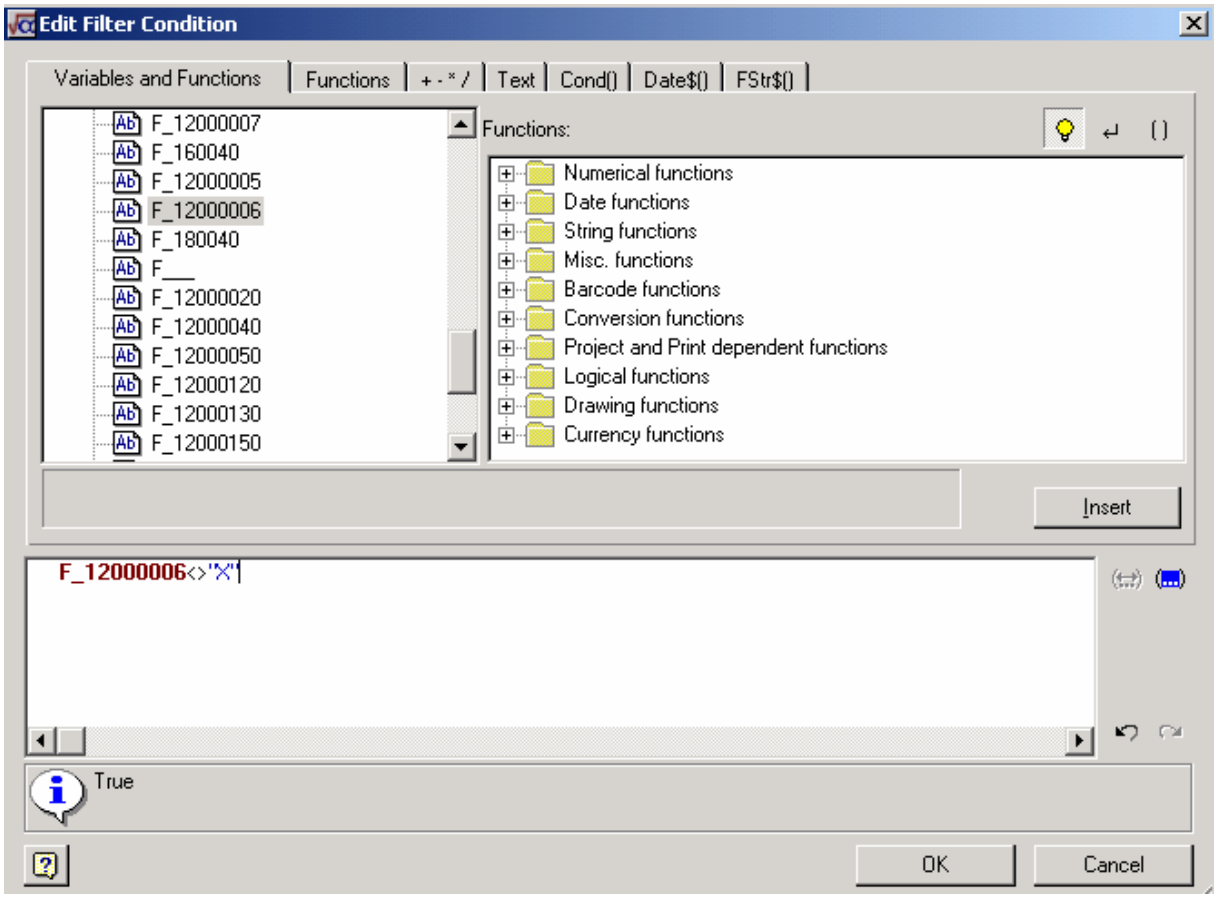
QQ.3.6.b. LIST OF PARTS WITHOUT TERMINALS

Condition: All the terminal strips belong to the same article group, for example to X.

1. If the fields for the list template are determined (for List of parts, simple), define the filter using **Project** menu □ **Filter**.
2. Double-click the "***Fields***" folder in the left upper area and double-click the field code you wish to define a filter for. In the example for the article group this is the code F_12000006.

The field code is transferred into the bottom area of the window.

2. Type the expression
3. F_12000006 <> "X"
   in the bottom area of the window.



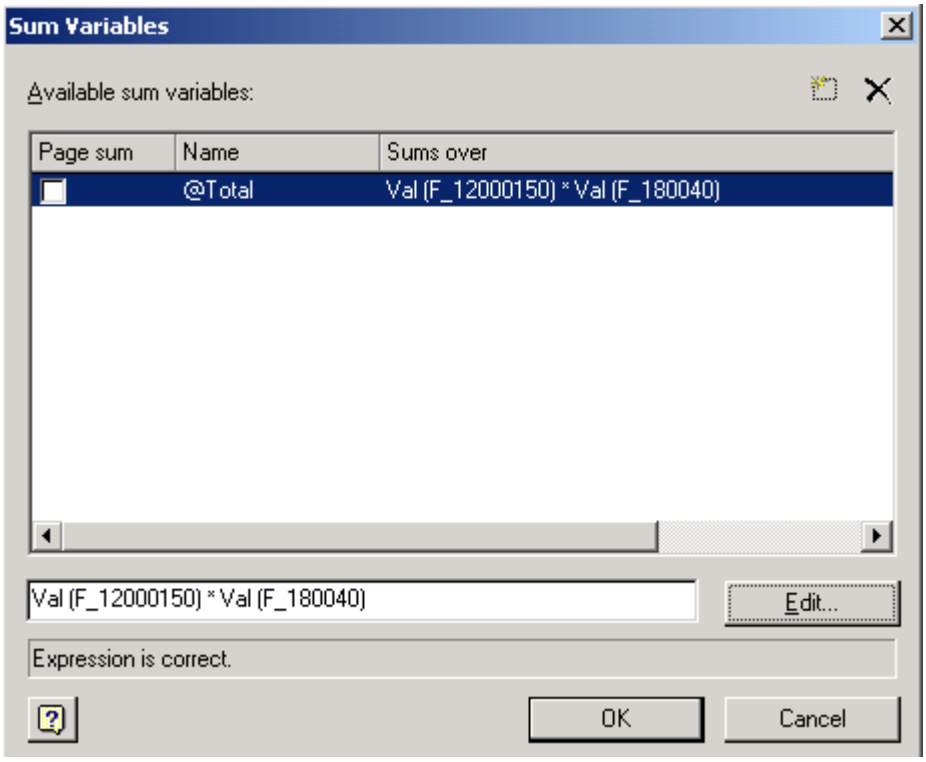
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

QQ.3.7. CALCULATIONS

To execute calculations, for example about costs of an order, it is necessary to define sum variables. You can define sum variables by selecting **Project** menu **Sum variables**.

QQ.3.7.a. CALCULATION OF ORDER COSTS

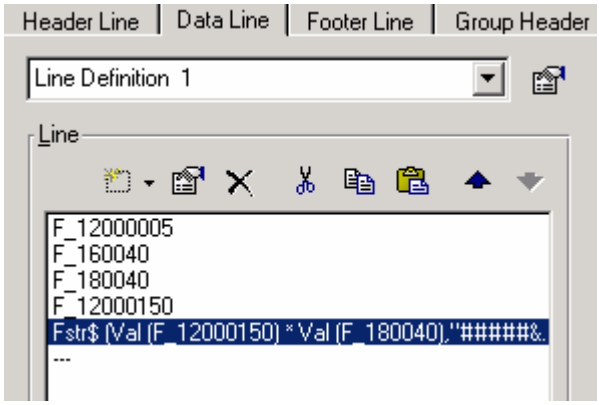
1. Define sum variables
   Calculate the total price of the articles of a type first. The total price is calculated as follows:
   Unit price (code 12000150) **x Number (**code 180040).
   The sum variable "Total" is defined as follows:



   As the unit price and number are transferred from the *SEE Electrical* project as text (=character string), the contents of the field is converted into a number using the **Val** command**.**
   The option "***Page sum***" must not be activated, as otherwise the prices in one page will be accumulated. However, the total price of all the articles in the list will not be calculated.

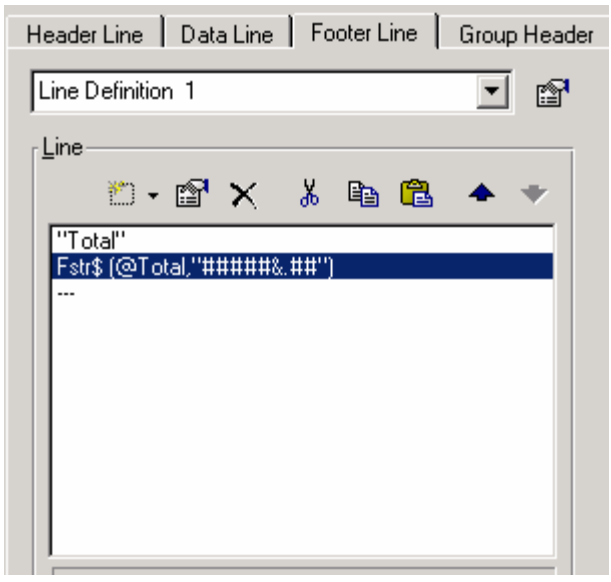
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

2. Edit data line
If the sum variable is defined, the table can be edited. The data line includes the record **Unit price x Number** in addition to the data fields.



The result of the calculation **Unit price x Number** is a number. This number must be formatted (2 decimal places) by using the command:
Fstr$(<value>,"#####.##").
in the form

3. Edit Footer line
The total for the whole project is recorded in the footer line:



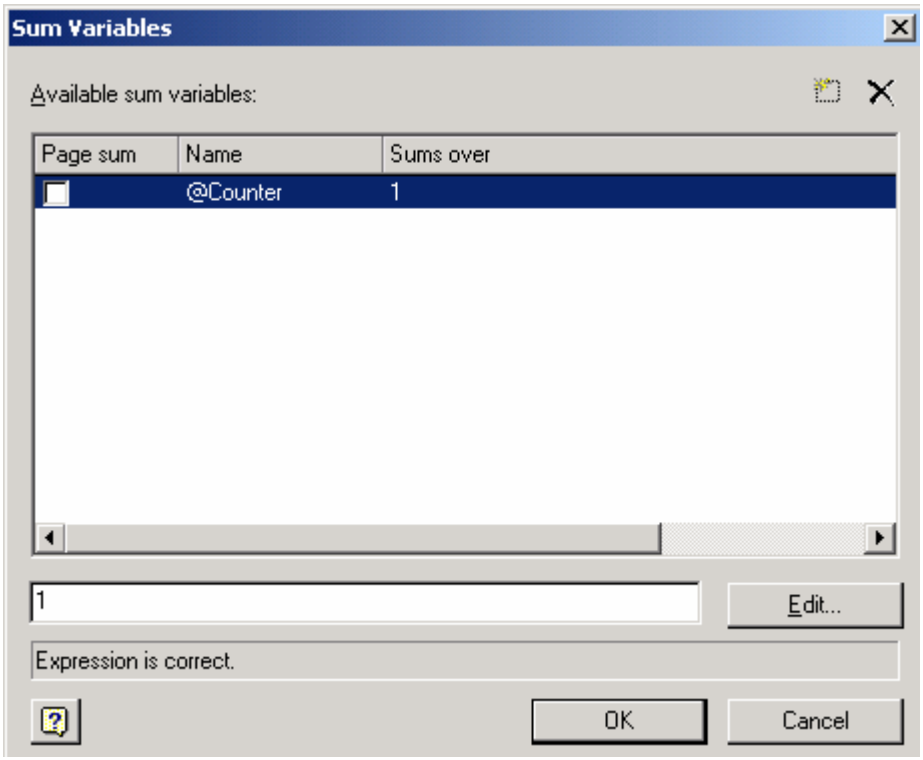
The number is formatted by using the command: **Fstr$(<value>,"####&.##")**.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

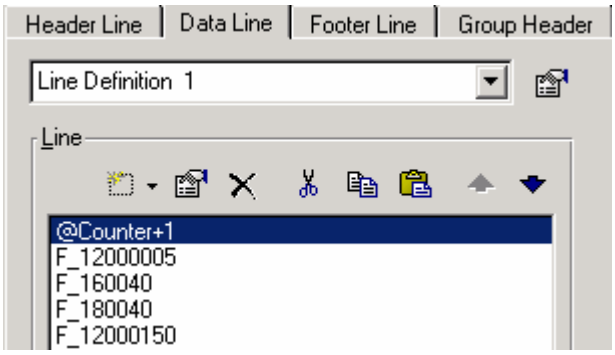
QQ.3.8. LINE NUMBERING

If you wish to print a current number per line, each line number must be calculated.
Define a sum variable for the line numbers counter as described in the chapter "*Calculations*" using "Project" -> Sum variables.

1. Define sum variable



2. Edit data line
After you define the sum variable, the table can be edited. The data line includes another record **<counter>+1** in addition to the data lines.



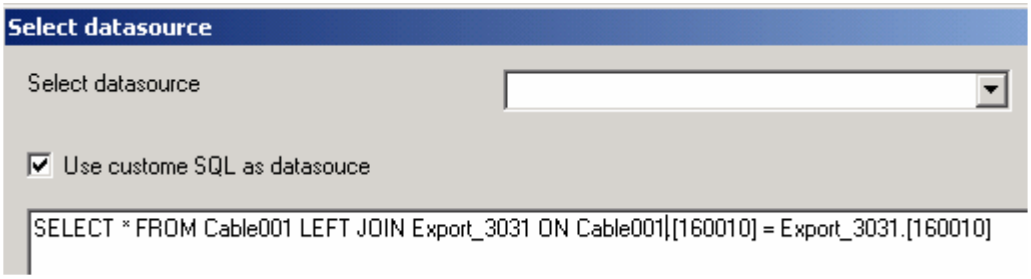
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

QQ.4. DEFINING SQL-QUERIES FOR LISTS AND LABELS

SQL-queries allow generating lists that go beyond the simple ordering, filtering and grouping fields from a database list.

QQ.4.1. JOINING TWO LISTS

It is possible to join data from two different database lists into one common list using *SQL*-queries in the ***List and Labels*** area.


Select datasource
Select datasource
Use custome SQL as datasouce
SELECT * FROM Cable001 LEFT JOIN Export_3031 ON Cable001.[160010] = Export_3031.[160010]

QQ.4.1.a. JOINING INFORMATION FROM CABLE LIST AND CABLE-WIRES LIST

1. Define a SQL-query
   SELECT * FROM Cable001 LEFT JOIN Export_3031 ON Cable001.[160010] = Export_3031.[160010]

Example for SQL-query:

SELECT	The SELECT statement selects data from a table. ***Syntax of the statement***: SELECT <fields> FROM <table in Access database>
*	All of the fields are selected.
LEFT JOIN	The second table is joined to the first table.
ON <value1> = <value2>	It defines which values in the two tables have to match in order to join data. In the example above, it is the component name (code 160010). The values must be defined in the form <table name>.[field name].

2. Defining Group header
   Data within a list can be structured by inserting group headers.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Example:

The cable data is displayed in a group header area above the cable-wires data.

The definition of the group header is made in 3 lines:

Cables — **List Header**

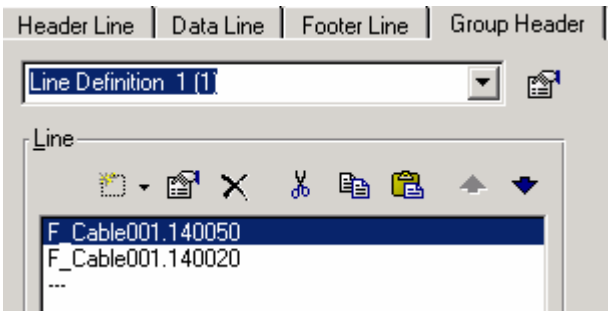
Cable: W1 **Type**: NYY 5x1,5qmm **Length**: 20m — **Group header**

Nr.	Colour	Size	Target 1	Target 2	*Group header*
1	sw	1,5	X1:L1	1M1:U	**Data line**
2	sw	1,5	X1:L2	1M1:V	**Data line**
3	sw	1,5	X1:L3	1M1:W	**Data line**
5	gn/ge	1,5	X1:PE	1M1:PE	**Data line**

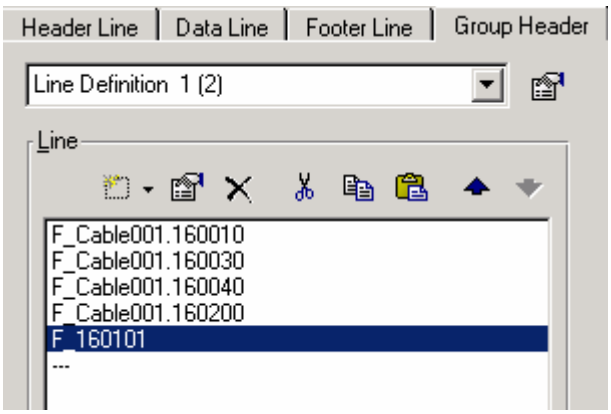
Cable: W2 **Type**: NYY 5x1,5qmm **Length**: 10m — **Group header**

Nr.	Colour	Size	Target 1	Target 2	*Group header*
1	sw	1,5	X2:L1	2M1:U	**Data line**
2	sw	1,5	X2:L2	2M1:V	**Data line**
3	sw	1,5	X2:L3	2M1:W	**Data line**
5	gn/ge	1,5	X2:PE	2M1:PE	**Data line**

Page 1 — **Footer**

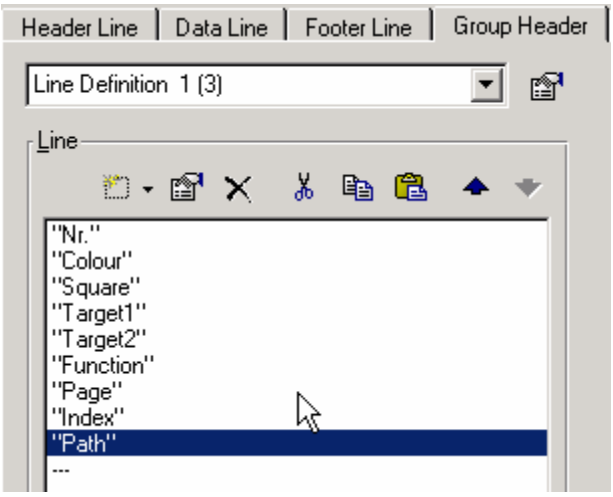


Cable Function - location



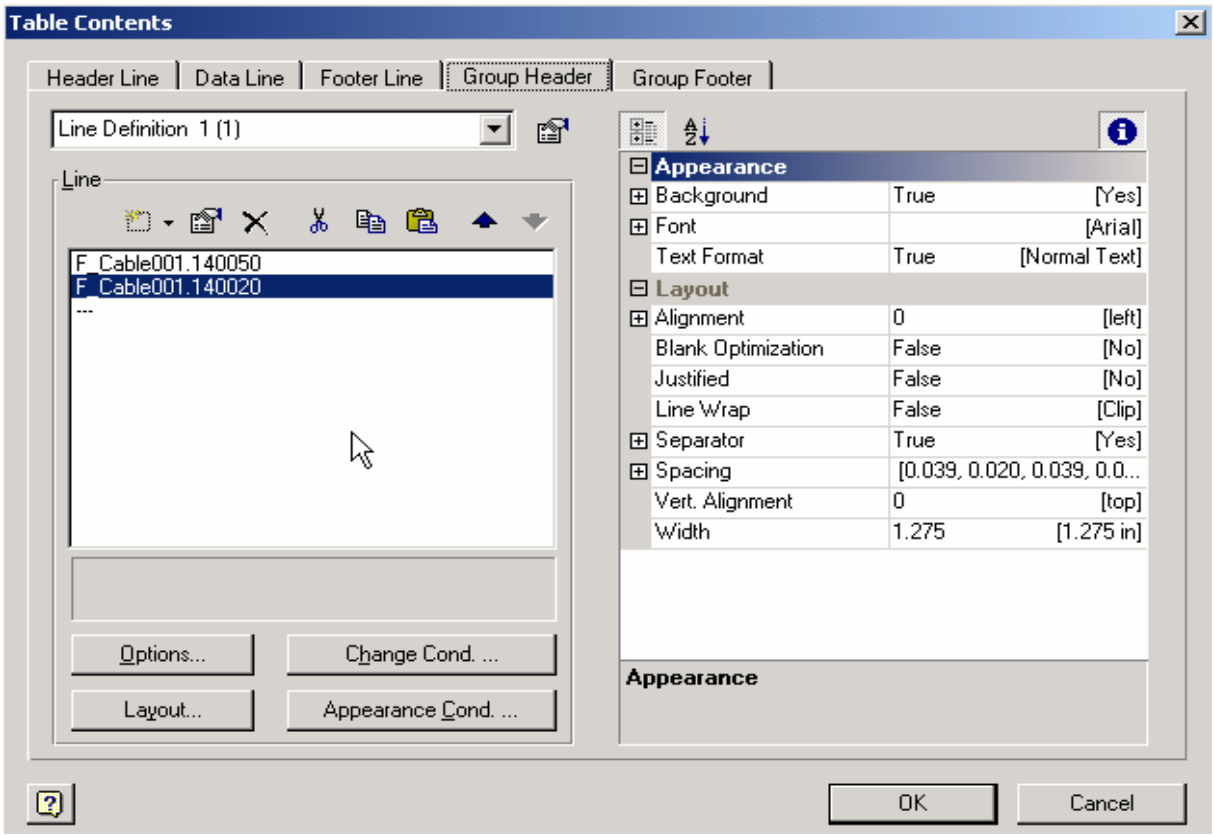
Cable name, type, length ...

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



Column titles

2.a Insert a page break after each group
It is essential in some cases to continue on a new page after fulfilling a specific condition, although the previous page has not been finished yet. If you work with a group header, you can define the condition for your list, too.



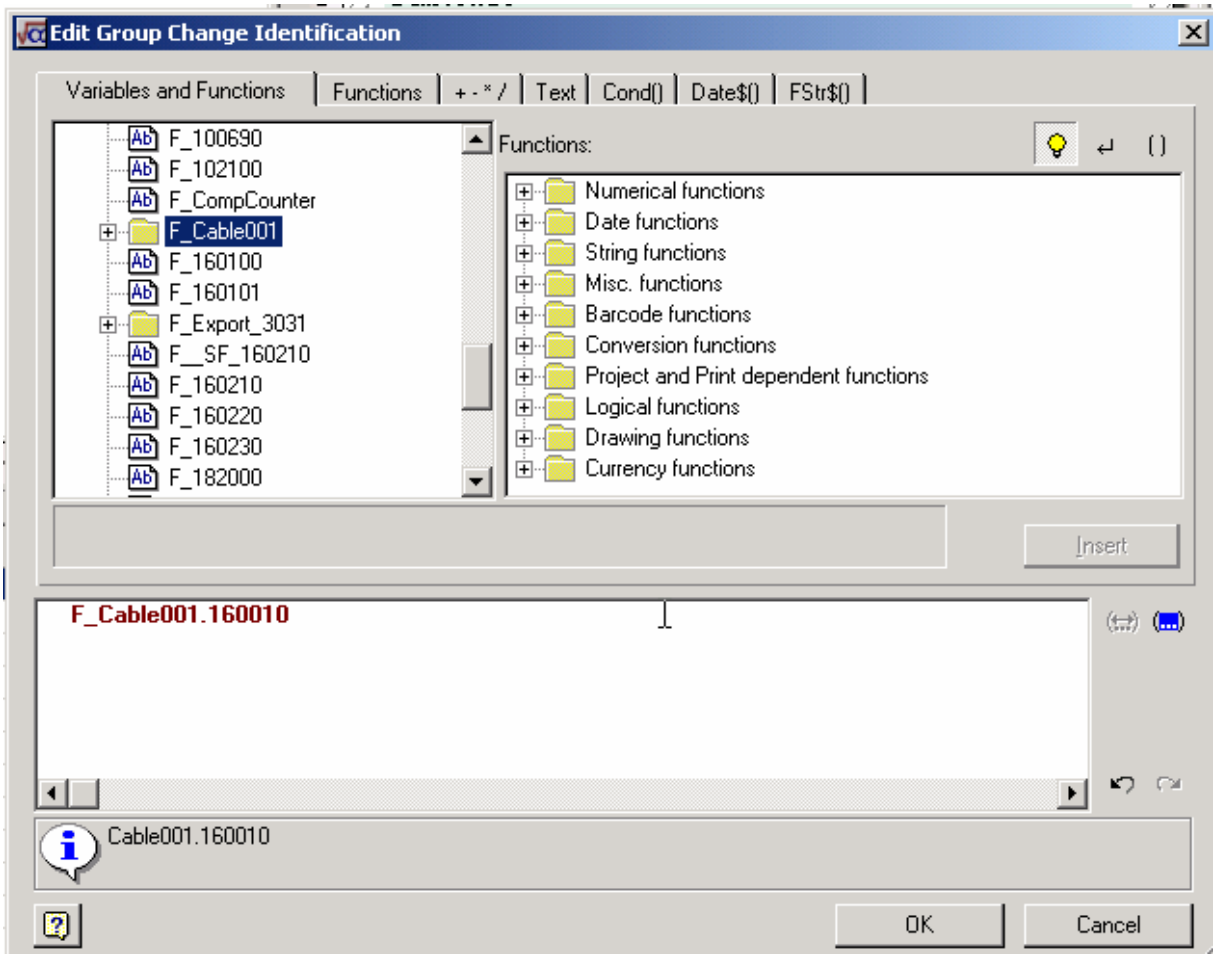
Select

to type the condition.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Select

and define page break after group changing:



The group must change when the cable name changes (field code 160010) that comes from table "Cable001".

QQ.4.2. FILTERING DOUBLE RECORDS

You can filter double records using the *SQL*-statement SELECT with the predicate DISTINCT.

Example: A list including functions and locations on the pages must be created.

Each combination must be displayed only once.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

QQ.4.2.a. LIST OF FUNCTIONS/LOCATIONS ON PAGES

1. Define a *SQL*-query
SELECT DISTINCT Export_3001.[180015], Export_3001.[180018] FROM Export_3001

SELECT DISTINCT	The SELECT statement selects data from the table. Each combination of the selected data is displayed only once by using the predicate DISTINCT. Syntax of the statement: SELECT <fields> FROM <Table of Access database> As the fields here do not have names, enclose the field code in square brackets "[" and "]".

2. Place fields for function and location.

QQ.4.3. SORTING A LIST

To sort lists in a specific way, use *SQL*-statements.

Examples:

If you want to receive an order list from the part list, the list has to be sorted by manufacturer.
If the lengths of equal cables must be added, the list must be sorted by cable type before addition.

QQ.4.3.a. ORDER LIST

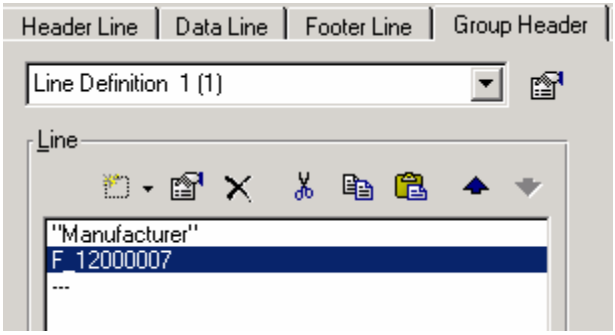
If you want to receive an order list from the part list, the list must be sorted by manufacturer/manufacturer. Then the articles must be grouped beneath the manufacturer. It might be necessary to break the page, if the manufacturer/manufacturer changes.

1. Define a *SQL*-query
SELECT * FROM Export_3100 ORDER BY Export_3100.[12000007]

SELECT	The SELECT Statement selects data from a table. ***Statement syntax***: SELECT <fields> FROM <table of Access database>
ORDER BY	This clause enables sorting of lists. ***Syntax***: ORDER BY<value> The value must be defined under the following form: <table name>.<[field name]>

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

2. Define a Group header



2a. Define group change identification

Select

, the group change identification is

.

Select then

and specify to start on a new page, if the group changes.

QQ.4.3.b. CABLE LENGTH ADDITION

Add the length of cables of equal types.

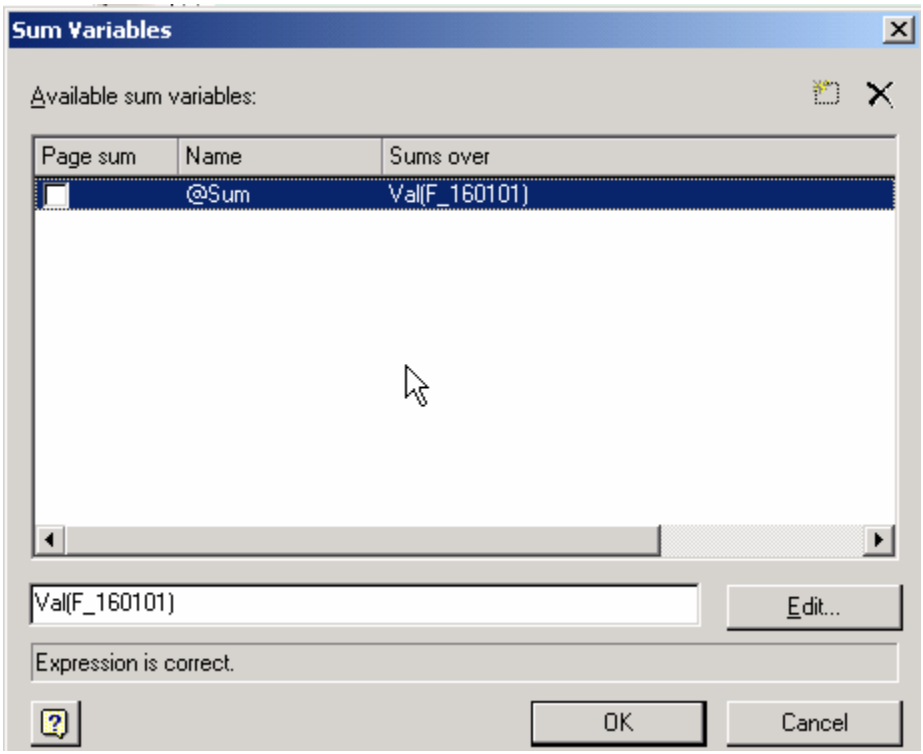
1. Define a *SQL*-query
SELECT * FROM Cable001 ORDER BY Cable001 [160040]
SQL-query:

SELECT	The SELECT Statement selects data from a table. ***Syntax***: SELECT <fields> FROM <table in Access database>
ORDER BY	To sum the length of cables of equal types, sort the list by cable type first. Use the ORDER BY clause. To sort then the cables by name, specify the name as second parameter. ***Syntax***: ORDER BY <value> The value must be defined under the following form <table name>.<[field name]>. Multiple sorting criteria must be separated with commas ",".

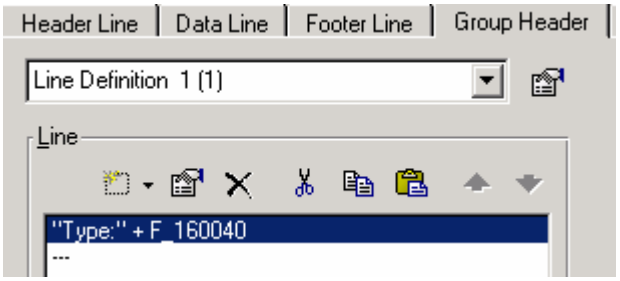
2. Define Sum variables
To calculate the cable length, define a sum variable.
You can define sum variables using **Project menu** □ **Sum variables**

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

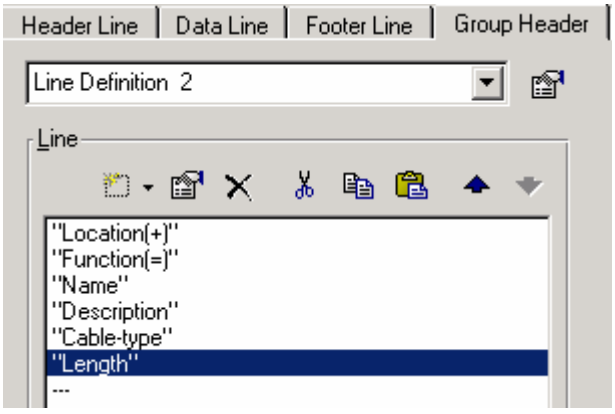
Example: Calculate cable length



3. Specify a Group header



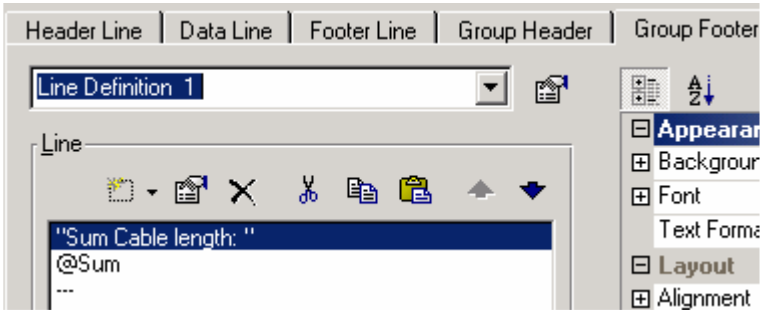
Line 1



Line 2

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

4. Data line must include the cable length.
5. Define a Footer line
Calculation of cable length for multiple cable types is performed in the Group Footer.



6. Define the group change identification:

QQ.4.3.c. LENGTH CALCULATION FOR CABLE CHANNELS AND RAILS

1: Define a *SQL*-query

SELECT * FROM Export_3010A, Export_3011 WHERE Export_3010A.Name=Export_3011.[160010] ORDER BY Export_3010A.ObjectType, Export_3010A.Name

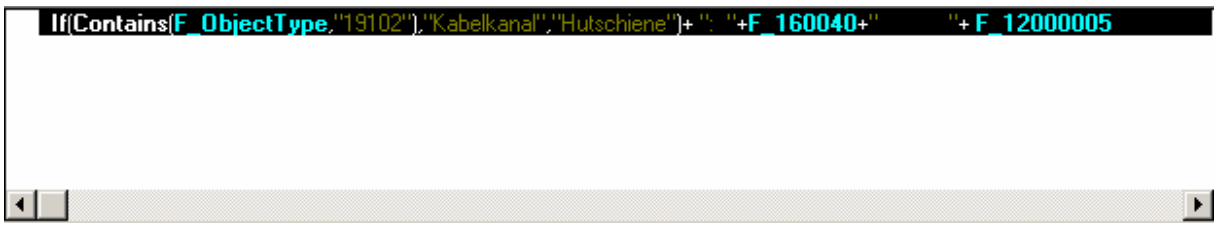
Clauses *SQL*-query:

SELECT	The SELECT Statement selects data from a table. ***Syntax***: SELECT <fields> FROM <table in Access database> Multiple tables are separated with commas ",".
WHERE	WHERE <value1> = <value2> You can define which values from the two tables must be equal, in order to calculate data. In this example, the values are component name in the NAME field of the Export_3010A table and component name in the 160010 field of the Export_3010A table. The values must be defined under the following form: <table name>.[field code].
ORDER BY	To perform printing of the cable channels first and rails after them, the sorting by object types is first necessary. Use the ORDER BY clause. Then, sorting by type must be done before sorting by component name. ***Syntax***: ORDER BY<value> The value must be defined under the following form: <table name>.[field code]. Multiple sorting criteria must be separated with commas ",".

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

2. Define sum variables. Compare the "Cable length addition" example and follow the steps there.
3. Define group header. It contains the record:

If(Contains(F_ObjectType,"19102"),"Cable channel", "Rail")+ ": "+F_160040+" "+ F_12000005



The expression defines to print the "*Cable channel*" text for objects of type19102 only, and otherwise – the text "Rail".

Syntax

If (Contains (<field>,"value"),
<"Result, if condition is complied with">,
<"Result, if condition is not complied with (can be omitted)">)
<Field 1> + ": "+<Field 2>

This is also a possibility to join different fields in one line.

4. The data line must contain the length; compare the "Cable length addition" example.
5. Define a Footer Line as in the "Cable length addition" example.
6. Define group change identification as in the "Cable length addition" example but the change must be executed by changing the object type, i.e. the condition is F_ObjectType

QQ.4.4. MULTIPLE PRINTING OF LABELS

If you need to print labels multiple times, print all the labels first and then once again, or print the labels twice over.

1. Print multiple labels (List of products)
   SELECT * FROM Export_3010 UNION ALL SELECT * FROM Export_3010
   i.e. the SELECT-statement is repeated after UNION ALL.
   The statement:
   SELECT * FROM Export_3010 UNION ALL SELECT * FROM Export_3010 UNION ALL SELECT * FROM Export_3010
   executes three times repeated printing.
2. Print multiple labels serially (List of products)
   SELECT * FROM Export_3010 ORDER BY Export_3010.[160010] UNION ALL SELECT * FROM Export_3010 ORDER BY Export_3010.[160010]
   The same labels appear directly one after another, for example
   S1 S1 F1 F1
   Q1 Q1 M1 M1

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

3. For Labels for terminals (List of spare parts), it is necessary to create a link to the List of products in order to perform filtering on the components in the circuit diagram.
SELECT * FROM Export_3101 LEFT JOIN Export_3010 ON Export_3101.[160010]=Export_3010.[160010] ORDER BY Export_3101.[160010] UNION ALL SELECT * FROM Export_3010 LEFT JOIN Export_3010 ON Export_3101.[160010]=Export_3010.[160010 ORDER BY Export_30101.[160010]

QQ.4.5. LABELS FOR COMPONENT NAMES WITH DEFINED CONTENT

It is possible to print labels only for defined component name content as follows:

1. Only records with defined content
SELECT * FROM Export_3010
WHERE (Instr (Export_3010.[160010],"K") OR Instr (Export_3010.[160010],"S"))
2. Only records without defined content
SELECT * FROM Export_3010
WHERE NOT (Instr (Export_3010.[160010],"M") OR Instr (Export_3010.[160010],"Q"))

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

QQ.5. USEFUL TABLES IN ACCESS DATABASE

AllTypesDISTINCT	All types in the project (without types, incl. subtypes) –no data about article's information registry –without number
Bom_Explode2	All types in the project (without types, incl. subtypes) –no
Bom01	data about article's registry of information –numbers.
Cable001	List of cables
ComponentTypes	Component names and types
ComponentTypesExploded	Component names and types
Export_3001	List of documents
Export_3010	List of products with components from Circuit diagrams and Cabinets (channel, top hat rail, etc .)
Export_3011	List of products with type information
Export_3020	Terminal names and types as well as article's registry of information
Export_3020A	List of terminals without type information
Export_3020B	List of terminals with type information
Export_3020D	All terminal types – article's registry of information –without number
Export_3025	Connector List
Export_3025	Connector Pin List
Export_3030	List of cables
Export_3030A	
Export_3030B	Only cable types
Export_3030C	Only different cables types
Export_3031	List of cable-wires
Export_3040	List of contacts
Export_3050	List of PLC
Export_3060	List of wires
Export_3070	List of potentials
Export_3100	List of parts
Export_3101	List of spare parts
Export_3102	List of parts, simple
Export_3103	List of spareparts, simple
Export_3104_Terminals	Combination of list of terminals and list of wires
Export_3180	List of products without components from Cabinets and less columns
Export_3181	Terminal list with x-y-coordinates and type
Export_3182	List of cable-wires with x-y-coordinates
Export_3225	List of Aspect Functions
Export_3226	List of Aspect Locations
Export_3280	Multicores
Export_3285	Multicores-wires
Export_Multiref	List of products without Cabinets with type Id
TypeInformation	Type Information

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

QQ.6. STATEMENT SUMMARY AND COMMANDS IN LISTS AND LABELS

QQ.6.1. SQL STATEMENTS

QQ.6.1.a. SELECT

The SELECT statement selects data from a database.

Syntax:
SELECT <fields> FROM <table in Access database>
Fields
The fields might be explicitly defined under the form:
<table name, containing the field>.<field name>
If you know the "right" field name, you can type it directly behind the dot operator. The field codes must be placed between square brackets "[" and "]".
Multiple fields can be separated with commas ",".
Select all of the fields in the table using an asterisk (*).

Examples:
SELECT ComponentTypes.Device, ComponentTypes.Location
or
SELECT Export_3001.[180015], Export_3001.[180018]
or
*SELECT **

Table in Access database
You can type the name of one or more tables where the data comes from. Multiple tables are separated with commas ",".

Example.
*SELECT * FROM Export_3010A, Export_3011*

Combine information with the help of SELECT:
Example for generation of combined result:.

```
SELECT Export_3010.[140050]+Export_3010.[140030]+
Export_3010.[160010] AS NameCombined,
Export_3011.[Manufacturer]+ Export_3011.[Goodsgroup]
AS Articledata
FROM Export_3010 LEFT JOIN Export_3011
```

QQ.6.1.b. SELECT DISTINCT

The SELECT statement with the DISTINCT predicate allows you to avoid double records.

Syntax:
SELECT DISTINCT <fields> FROM <table in Access database>

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Example:

```
SELECT DISTINCT Export_3001.[180015], Export_3001.[180018]
FROM Export_3001
```

QQ.6.1.c. JOIN –CLAUSE

Databases usually contain a lot of tables. JOIN-clause is used for joining tables. It enables you to list data that cannot be found in one table.

Example: The list of cables contains information about the length of the cables but the list of cable-cores does not contain such information. If a list must contain information about cables length and cable-cores, it must be composed by matching data coming from both lists.

You can achieve this by joining the records of both lists. You must define a SQL-statement, for example, with a WHERE or ON clause.

Syntax of WHERE clause:
WHERE <Condition for selection>
Condition for selection
<Value1> <relational operator> <Value2>
Values : Specify which values from both tables must match in order to join the data. For example, this can be the component name (field code 160010). The values must be defined in the following format: <table name>.[field name].
Relational operators: =, <, > are allowed

Example:

```
SELECT *
   FROM Export_3010A, Export_3011
     WHERE Export_3010A.Name=Export_3011.[160010]
```

Syntax of ON clause:
<table 1 in Access database> LEFT JOIN <table 2 in Access database> ON <Condition for assignment>
Condition for assignment
<Value1> = <Value2>
It defines which values from both tables must match in order to join the data. In this example, it is the component name (field code 160010). The values must be defined in the <table name>.[field name] format.

Example:

```
SELECT *
   FROM Cable001
     LEFT JOIN  Export_3031
       ON Cable001.[160010] = Export_3031.[160010]
```

The ON clause may include several fields. The fields must be linked by using logical operators:

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Example:

```
SELECT *
  FROM Export_3103
    LEFT JOIN Export_3010
      ON
          Export_3103.[140050]=Export_3010.[140050]
        AND
          Export_3103.[160010]=Export_3010.[160010]
```

QQ.6.1.d. ORDER BY

This clause allows sorting the list.

Syntax:
ORDER BY <value>
Value
The value must be defined in the following format: <table name>.<[field name]>.
Example:
SELECT * FROM Export_3100 ORDER BY Export_3100.[12000007]
Multiple sorting criteria are separated with commas ",".

QQ.6.1.e. WHERE/WHERE NOT

The WHERE clause can be used for joining two tables. WHERE and WHERE NOT can also be used for filtering.

Join two tables
See JOIN clause
Filtering

Some examples for using the WHERE clause:
Perform filtering of articles for one particular manufacturer from the parts list Syntax:
WHERE <value1> <relational operator> <value2>
Value1
The values must be defined in the <table name>.[field name] format.
Relational operators:
=, <, > allowed
Logical operators:
and, or, not
Value2
The values must be defined under a "<text>" form.

Example:

```
SELECT * FROM Export_3100
 WHERE Export_3100.[1200007]="Siemens"
```

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Print labels only for components with specified name value

Syntax:
WHERE (Instr (<Field>,"value"]
Field
The field from the project database must be defined in the following format: <table name>.[field name].

Value
Enter text for the field value.
Logical operators:
and, or, and not

Examples:
1. Only records with defined values (component name "K" or "S") must be selected
SELECT * FROM Export_3010
WHERE (Instr (Export_3010.[160010],"K") OR Instr (Export_3010.[160010],"S"))
2. Select only records with the defined values (component name without "M" or "Q")
SELECT * FROM Export_3010
WHERE NOT (Instr (Export_3010.[160010],"M") OR Instr (Export_3010.[160010],"Q"))

QQ.6.1.f. UNION ALL

UNION ALL can be used to execute a query in the same list twice or multiple times. (For example, it can be used for multiple printing of labels).

Syntax:
SELECT <selection> UNION ALL SELECT < selection>
Example:
SELECT * FROM Export_3010 UNION ALL SELECT * FROM Export_3010
(double printing)
or
SELECT * FROM Export_3010 UNION ALL SELECT * FROM Export_3010
UNION ALL SELECT * FROM Export_3010
(triple printing)

VAL

VAL converts a text into a number.

Syntax:
Val (<value>)

Example:
SELECT * FROM Export_3030A WHERE (Val(Export_3030A.[160200]) > Val("4.00")) and (Val(Export_3030A.[160200]) < Val("8.00")

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

QQ.6.1.g. UPPERCASE/ LOWERCASE

It is not important for the SQL-query whether upper case or lowercase is used in the *SEE Electrical* database.

Example:
SELECT * FROM Export_3100 WHERE Export_3100.[12000005]="COIL"
and
SELECT * FROM Export_3100 WHERE Export_3100.[12000005]="Coil"
provide the same result, i.e. all records containing the "Coil" text are provided whether written in uppercase or lowercase.

QQ.6.1.h. LOGICAL OPERATORS

The operators AND, OR, and NOT are available.

QQ.6.1.i. RELATIONAL OPERATORS

The operators =, <, >, <> are available. InStr expression is available, too.
InStr
InSTR checks whether the string in the defined field contains a pattern string.

Syntax: InStr (<field>,"<pattern string>")

Example: Component names must contain the letter "M"
WHERE (InStr (Export_3010.[160010],"M"))

QQ.6.2. COMMANDS IN LISTS AND LABELS

QQ.6.2.a. CONDITIONS: IF

Syntax
If (Contains (<field>,"value"),
<"result, if the condition is satisfied">,
<"result, if the condition is not satisfied (can be omitted)">)

Example: Print labels (in one line, it doesn't match more than 4 characters), a page break can occur, but only when an empty character is found.
(F_160010)>3,Left$ (F_160010,3)+" "+Right$ (F_160010,4),F_160010)
If the component name is > 3 characters long, it is split into 3 characters on the left and 4 characters on the right with one space between. If the component name is <3 it remains whole.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

QQ.6.2.b. UPPER$/LOWER$

Upper$ (....) = "<VALUE>"
This command converts the letters case into upper so that it is possible afterwards to be compared with the value after "Upper$ ()" (this value must always be written in uppercase).
The LOWER command changes the letters case into lower.

QQ.6.2.c. VAL

Val(<field>)
VAL transforms a text into a number. The fields in *SEE Electrical* databases contain generally numbers as texts.

QQ.6.2.d. FSTR$

Fstr$(<value>),"####.##")
This command allows formatting numbers.

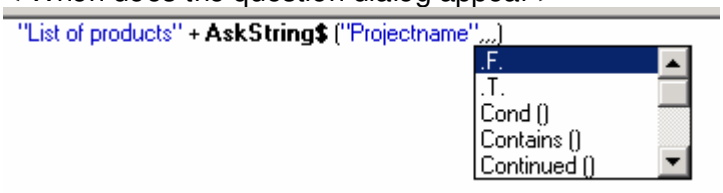
QQ.6.2.e. ASKSTRING$

AskString (<variable name>,<when does the question dialog appear>,<suggested value>,<max. number of characters>)

The variable "AskString {...} allows you to execute a dialogue with a text question and enter user-defined texts while creating the list.

Arguments:
<variable name>
This text will be shown when the question appears. The text must be placed between double quotation marks " .

< When does the question dialog appear >



By selecting .F., you specify that the question dialogue is executed only once – at the beginning of the list.
<suggested value>

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Enter the value to be automatically suggested. The text must be placed between double quotation marks ".
<max. number of characters>
Enter the max. number of characters expected for your text.

QQ.6.2.f. LEN, LEFT$, RIGHT$

Len gives the length of the strings.

Right$ or Left$ cut the right or the left characters from a string.

The following expression divides the strings into 2 parts if the length of strings exceeds a determined length. Using this, you can, for example, split the component name and print on two lines although a line break is not possible within one "word". Line break is possible only where a Blank character is.

If (Len(F_160010)>3,Left$ (F_160010,3)+" "+Right$ (F_160010,4), F_160010)

QQ.6.2.g. RELATIONAL OPERATORS

The operators =, <, >, <> are available. The expression "Contains" is available, too.

Example for Contains:
If(Contains(F_ObjectType,"19102"),"Cable channel","Rail")
This expression defines: for object type 19102 print the text *"Cable channel",* otherwise print the text *"Rail".*

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

RR CABINET LAYOUT

You can construct a *Cabinet Layout* as a part of the circuit diagram. It is possible to use symbols there, too.
Functions for dimensioning are provided with all levels of *SEE Electrical*.

RR.1.DRAWING CABINET LAYOUTS

Cabinet layouts can be created 1:1 or using a scale.

Drawing in 1:1 is convenient because you do not have to change the dimensioning scale; symbols for components are drawn directly 1:1 and they can be used again afterwards. The font size must be adapted. You need a page template with a standard sheet, where the cabinet layout fits 1:1. Change the dimensions in ***Page Properties***.

Working with a scale is convenient because you can use the A3 standard sheet later. The symbols must be created in another page using the 1:1 scaling. For symbols insertion, set the scaling factor. The dimensioning scale must be changed. Font size can be maintained.
Choose, in ***Page Properties***, whether to work with a scale or not.

RR.1.1. WITH THE CABINETS MODULE

(Standard)

The following exercises illustrate the creation of a cabinet scaled 1:10.

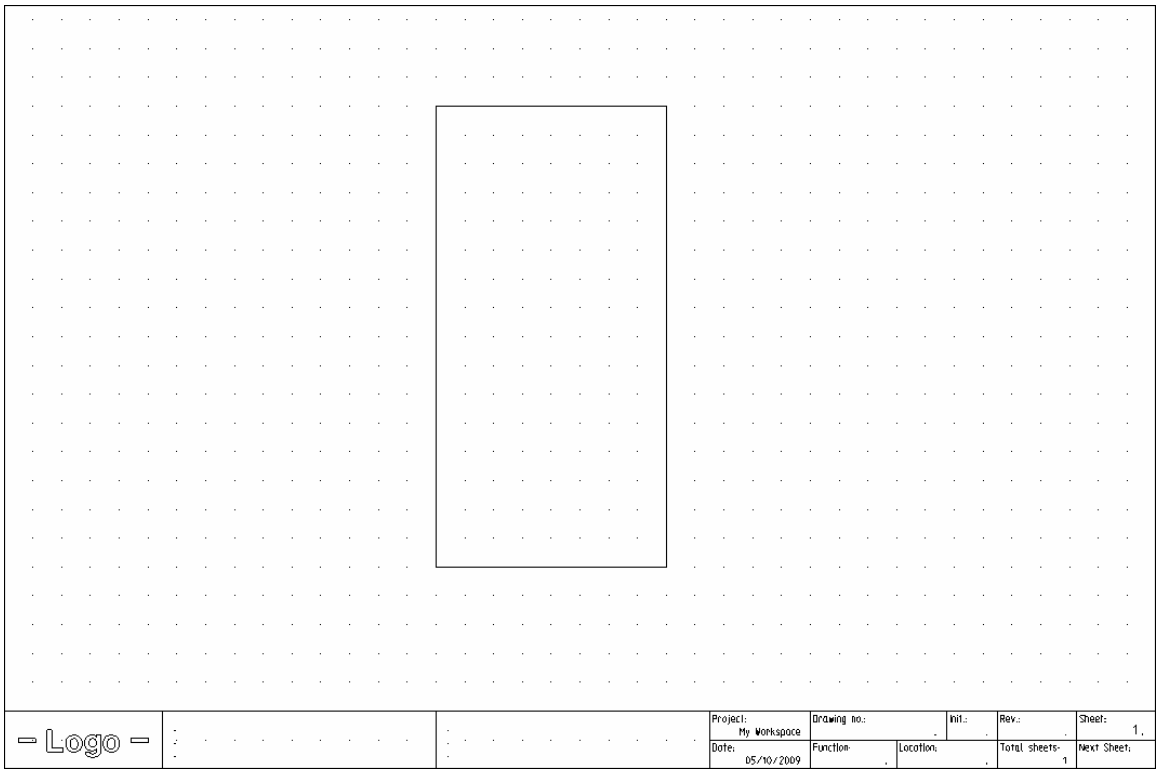
Exercise 40-1: Create a new page within the Cabinets module and load an appropriate page template.

- Create the new page in the known way.
- Select the *Cabinets* module in the ***Workspace Explorer***.

The default page template for Cabinets loads. The scale for drawing and the symbol scaling are 1:10 by default.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercise 40-2: Draw a panel 800 x 1600 mm.



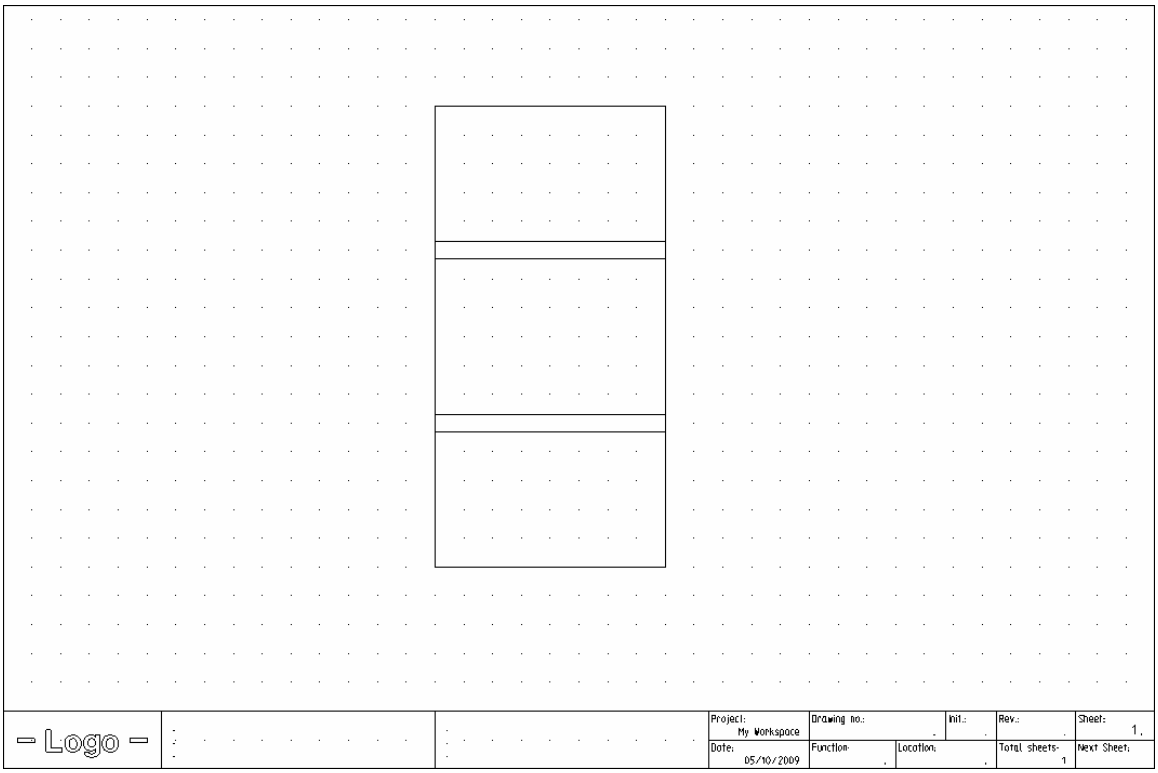
In the Status bar (to the right under the drawing area), the dimensions are shown while drawing lines and rectangles. Set a grid size (at least 10x10 mm).

You can draw a Panel.

1.CA **Cabinet**
2.CO **Panel** (**Elements** panel)
3.+ Select the starting point of the panel.
4.# Press the space bar.
5.> dX
6.# 800
7.> dY
8.# 1600

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercise 40-3: Draw a Cable/Wire Channel with a width of 60 mm and a length of 800 mm.



1.CA **Cabinet**
2.CO **Cable/Wire Channel** (**Elements** panel)
3.> Width
4.# 60
5.> Length
6.# 800
7.> **Insert Channel**
8.+ Place the cable channel in the desired place.
9. Place the cable channel again in another place.

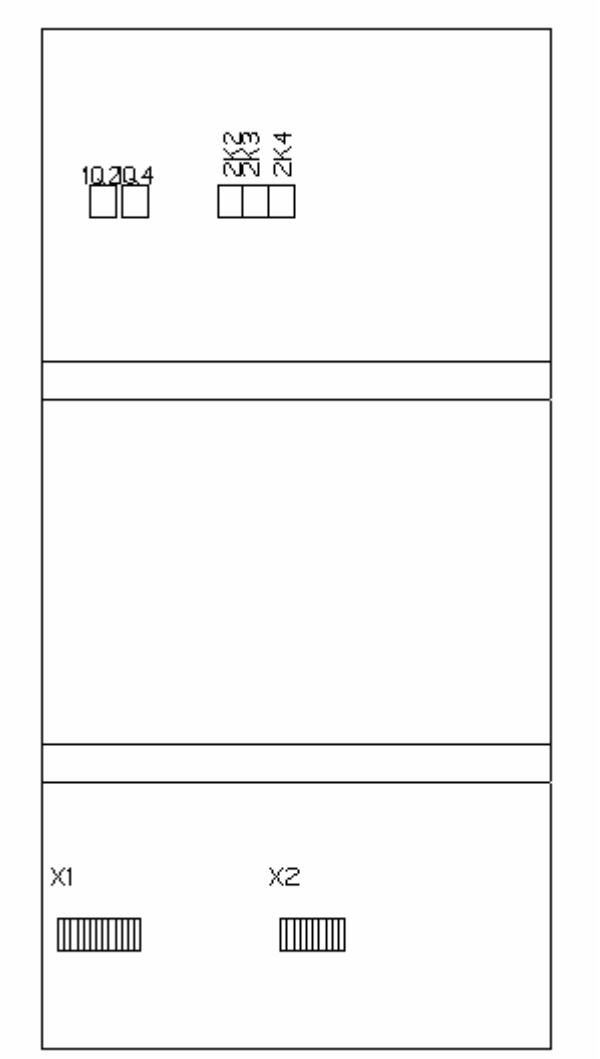
Through the "+" or "-" keys in the numeric block of the keyboard you can rotate the cable channels (or other elements) before placing them if you did not enter the correct angle when defining the width and length.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercise 40-4: Insert components using the pick list that contains all the components from the circuit diagram.

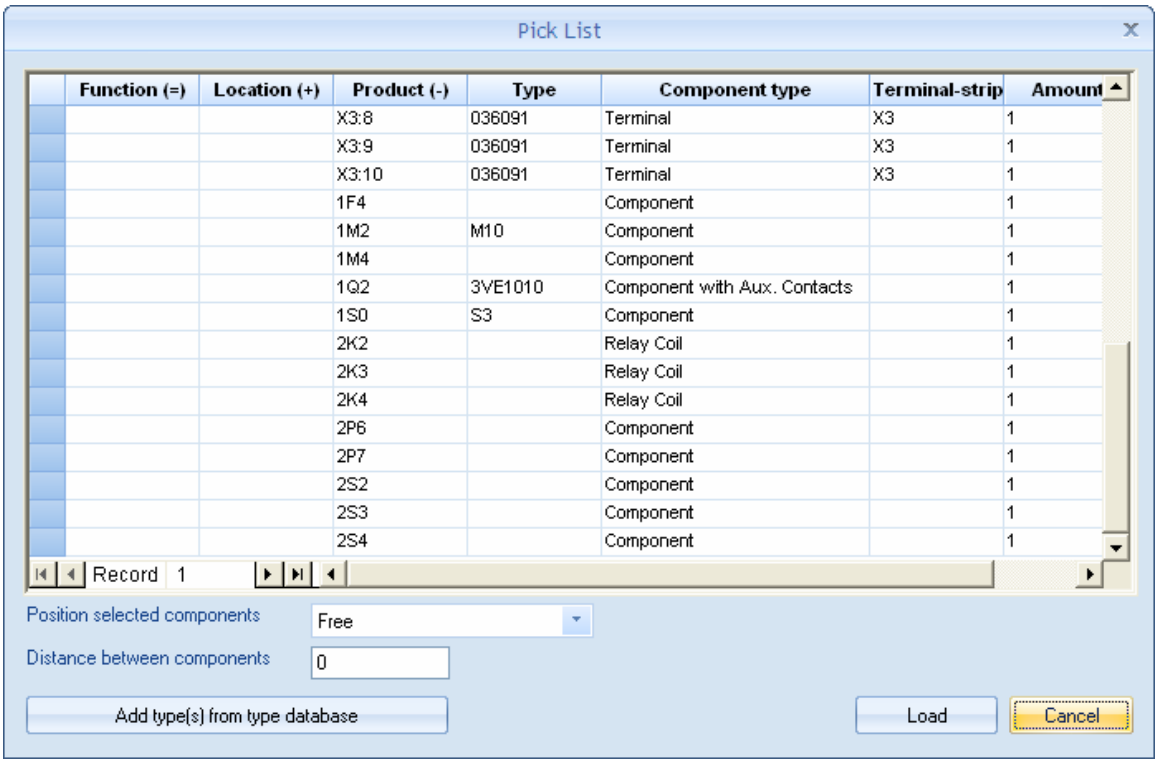
When you insert a component, it is deleted from the pick list. The component name appears automatically next to the symbol.

When you delete a component from the cabinet drawing, it appears in the list again.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

1.CA **Functions**
2.CO **Pick List** (**Other** panel)



All of the components in the circuit diagram are displayed in this list.

3.> Double-click a component to select it.
You can select several components using the standard *Windows* procedure for multiple selection. Click **Load** to insert the selected components into the cabinet drawing.
In the "***Position selected components***" field, you can choose between **free**, **horizontally** or **vertically** aligned for the selected components to be inserted on the page.
You can specify the distance between components. Afterwards, you can place the first component.
4.+ Place the first component.
Place the other components.

If you place a lot of devices horizontal or vertical - especially terminals - you might like to hide the texts of the symbols.

Pressing the h key while inserting them hides the component name.

Pressing H hides all component names and connection texts. The option is useful especially for terminals that are made by rectangles only.

Exercise 40-5: The relation between devices in Circuit diagrams and Cabinets is possible if you assign a type.

In the type database, you can set the "*Width*" and "*Height*" properties of the rectangle for the cabinet layout.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

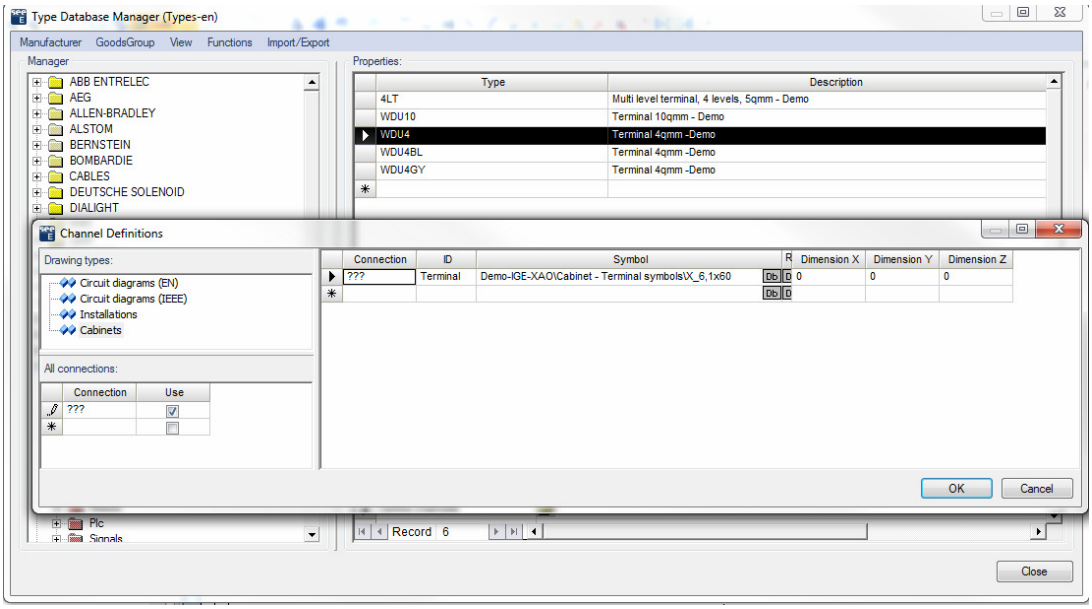
If width and height are not specified for a type, a rectangle identified with a yellow diagonal cross appears. This way you are warned that the size of this component is not real.

If you want a detailed view of your construction symbols, you can designe a graphic in scale 1:1 or you can import from the *DXF-DWG* format. The symbols for *Cabinets* must satisfy the same criteria as the symbols for *Circuit diagrams*, i.e. a symbol for relay coil must be assigned to the category of relay coils. You can assign symbols to a type in the type database in channel definition as shown in exercise below..

Exercise 40-6: Create a terminal symbol for the cabinet.

1. Create a new page within Cabinets.
   Create symbols on the new page.
2. Type the value 1 for scale and Symbol scaling as the symbols must be created with a scale 1:1.
3. Create a rectangle with a size 52x6.
4. Group the graphics into a Terminal symbol.
5. Select **Edit** **Text** **Edit Text** and change the size of the text for the component name to 35mm. As the symbol is minimized by the factor 10 after the insertion, it applies to texts too.
6. Move the text of the component name one position above the terminals (use the **Select single element** and **Move** commands)
7. Double click the symbol and switch off the visibility of all texts.
   The symbols for terminals are so small that the texts will overlap.
8. Insert the symbol into the MySymbols symbol database in the Cabinet folder. Type the Training-terminal name.
9. Go to the type database. Create a new terminal type, for example UK10-Training, assign the Symbol name for Cabinets property to it. In the Value column, enter the following: <Symbol database>\< Folder >\<Symbol name>, in our example MySymbols\ Cabinet \Training- terminal
10. If you have already placed terminals in the cabinet, delete them.
11. Assign the type UK10-Training to the terminals.
12. Place the terminals again.
    The symbol is now used.
13. For a terminal of a terminal strip, you can set to view the terminal strip name by double-clicking the terminal and activating the "*Show*" option for the component name.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



Exercise 40-7: Use several cabinet symbols in one component.

It is possible to insert cabinet symbols for all types of components in case more than one type of component has been added. In the channel definition, in the "***Cabinet symbol***" field, you have to define a symbol for the cabinet or the size of the rectangle or circle to be generated.
The sizes are defined in the following way:
33x30 (for a rectangle)
D30 (for a diameter)
or
R15 (for a radius).
If subtypes have been used, the symbols attached to them in the channel definition are also used.

If a component has multiple types and, in the channel definition, a cabinet symbol or a dimension has been defined for more than one for these types, all the symbols/rectangles for the component can be inserted in a cabinet. In this case you can select only one component and insert the single parts. There is no general rule how the other types have to be inserted in relation to the first one. If you select multiple parts and use horizontal or vertical automatic placing, the automatism is interrupted when you use a component with multiple symbols. You have to insert all the symbols for this component and after this for the next component, to define where to start the automatism again. Then the automatism continues.

Example
- if a coil consists of the coil itself and two add-ons, one to be mounted on left and the other to be mounted on right side, you have to insert first the add-on for the left side, then the coil and after that the add-on for the right side.
- if a coil consists of the coil itself and an add-on to be inserted on top of the coil, you must insert the coil first and then the add-on.

If you have to insert the first type of a component via the picklist and skip the others, the component is not present in the pick list any more (because the pick list contains only symbols inserted in the circuit diagram and not in the cabinet). Then you have to use the **Complete database component** command to complete the component.
If symbols for the cabinet are used, it is really important they have as much connection points as defined in the first line in the channel (where is defined the cabinet symbol).

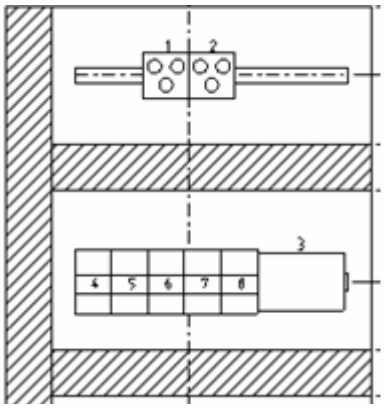
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

If you have to define a cabinet symbol, for example, in the first line in the channel definition and also in the second one, the one defined in the first line needs to contain as much connection points, as defined in the first line, and the cabinet symbol attached to the second line needs to contain as much connection points, as defined in the second line etc.

Hiding component names in case multiple symbols are used for a component
If you press the **H** key on the keyboard while placing the symbols, the component name is hidden for all the following symbols. If you want to display the name for the next symbol again, press the **H** key again.
It is possible to switch on or off the displaying of the name in the ***Component Properties***.

Exercise 40-8: By means of a section, delete again all components in your circuit diagram.

Exercise 40-9: Draw the rails in your cabinet.



1.K Cabinet
2.M Rail (in the Elements filed)
3.> Width
4.# 35
5.> Length
6.# 700
7.> **OK**
8.+ Place the rail.
9.+ Place the rail a second time and, if necessary, a third time.
10. Through the **Pick list** command, place components on the rails.

Exercise 40-10: Delete the components from the rails and move components from one rail to another.
Components are fixed on the rails, you cannot select them directly.
1. Right click.
The context menu appears.
2.M Select a single element in the component group.
3.+ Click the component to be selected.
4.+ Press the **CTRL** button if you want to select more components.
5. Delete the components.

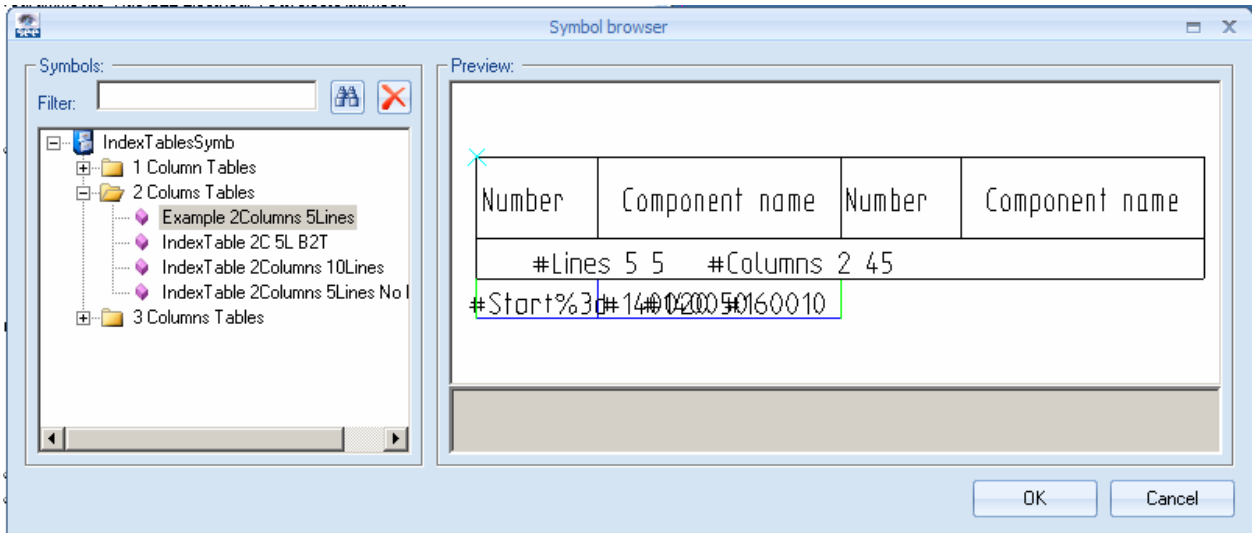
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

If you want to select more components, you can do this by the means of a section, as you do the following:

1.# Simultaneously, press the SHIFT and CTRL buttons on the keyboard.
2.+ Define the first point of the section.
3.+ Define the second point of the section.
4. Through the **Move selected elements** pop-up command, now you can move the components on another rail.

Exercise 40-11: Create index.
It is possible to insert an index table into the cabinet drawing.
The index is created with the help of user defined templates. The templates are located in the "IndexTableSymb.ses" symbol library by default.

If you activate the command on a page that has components on it, *SEE Electrical* offers all templates that are present:



Make your choice in the Symbol browser and click **OK**.
The Index table is generated and you have to insert it with the cursor.
If the number of components exceeds the number of the places available in the template, an error message appears. Delete the index and choose another template.
The index can be deleted like any other object.
If an index is already present in the drawing, the **Index components** command updates the existing index.
The component name is replaced by the ordinal number from the index. The component name is still present in the component, but it is automatically hidden.
In the "***Cabinet Properties***" there is a setting that allows you to define if each terminal is shown in the index table or only one entry for the whole strip.

Creation of templates for indexes:
The template is created using geometry and specific texts.
- #Lines <Number of lines> <Distance between lines>

This value sets number of lines and distance between lines, as well as Top to Bottom, or Bottom to Top direction of the drawing (negative distance -> Bottom to Top).
The text has to have the attribute "Normal". It can be placed at any position in the template.
Examples: #Lines 30 4 or #Lines 30 -4

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

- #Columns <Number of columns> <Distance between columns>

This value is only necessary, if you want to use more than one column.
The text has to have the attribute "Normal". It can be placed at any position in the template.
Examples: #Columns 2 36

- #Start <formatting attribute for ordinal number>

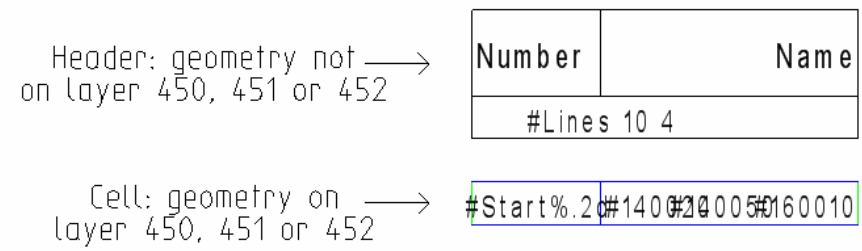
The text has to have the attribute "Normal". The %2d formatting attribute formats the ordinal number with minimum two digits, for ex. 01, 02, .., 10, …, 99, 100, ..

- Add texts with the text attribute "Normal" and with the component IDs you want to be displayed.

#160010 component name or terminal name/separator/number
#140020 component function
#140050 component location

- Add geometry.

You have to put the lines for the single cells that contain the index information on layers 450, 451 and 452.
The lines for the header must not be on these layers.



Block the geometry and the text as a symbol of "Cabinet, Index component table" type.
Store the symbol in the "IndexTableSymb.ses" library.

Exercise 40-12: Save your workspace.

RR.1.2. WITHOUT THE CABINETS MODULE

You will create the cabinet scaled 1:1.

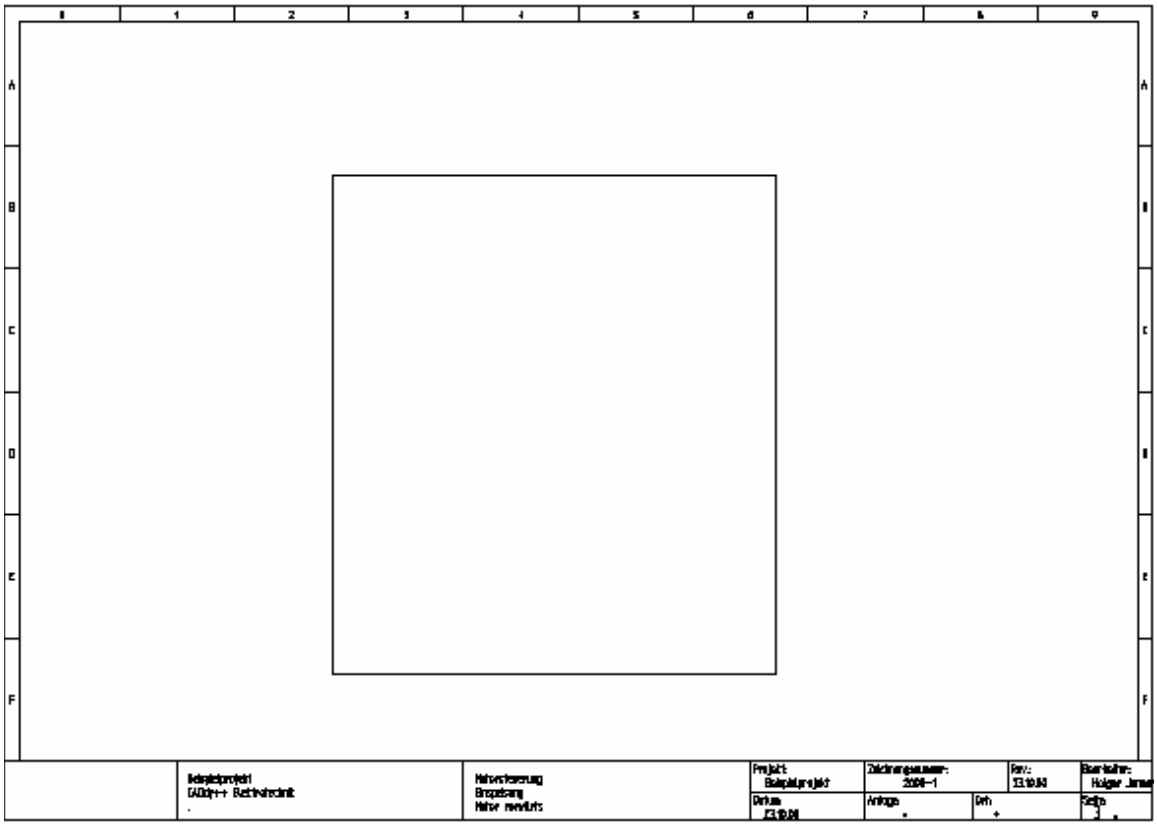
Exercise 40-13: Create a new page and load an appropriate page template.

1. Create the new page in the known way.
2. <Page>
   Choose the new page in the Workspace.
   Load the page template A3x5. It represents a page of A3 format enlarged by the factor 5.
   This page allows you to create a Cabinet with the maximum size of 1500 x 1250 mm.

2.CA **File**
3.CO **Open**
4.CA **Page template**
5.> <Your template>, for example A3x5
   The setting for scale and symbol are found in the page properties.
6.> **OK**
   The page template has been loaded. The workspace information and the page information have been saved in the standard sheet.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercise 40-14: Draw a panel 800 x 900 mm.



In the status bar (right beneath the drawing area), the dimensions are displayed as you are drawing the lines and rectangles.
Choose a larger grid through the Toolbars (at least 10x10 mm).)

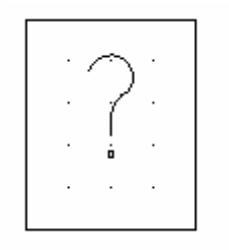
1.CA **Draw**
2.CO **Rectangle** (**Elements** panel)
3.+ Select the first point of the rectangle.
4.+ Select the second point of the rectangle.
Look at the dimensions in the status bar.

Exercise 40-15: Draw a rectangle 40 x 50 mm.

1.CA **Draw**
2.CO **Rectangle** (**Elements** panel)
3.+ Select the first point of the rectangle.
4.+ Select the second point of the rectangle.
Look at the dimensions in the status bar.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercise 40-16: Place one text for the component name.

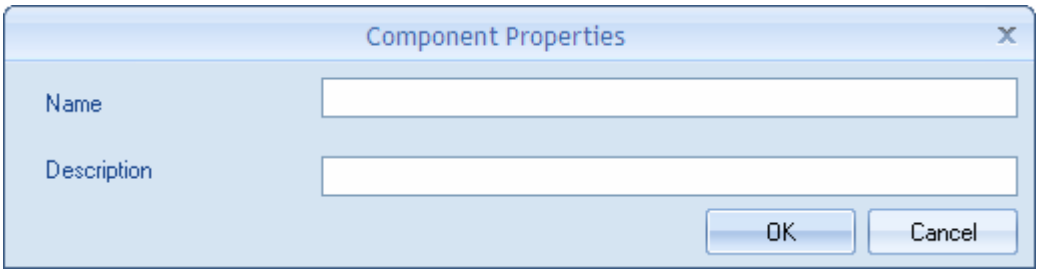


1.CA **Draw**
2.CO **New Text** (**Elements** panel)
3.> Height
Change the height for the text because the component and the text will be reduced when printing later. Text with height 3.5 would be too small.
4.# 35
5.> Centre justified
The text will be centred.
3.# ?
Type in a question mark "?" for the text.
4.+ Place the text within the component rectangle.

Exercise 40-17: Group the rectangle and the text into a symbol and save in the symbol database.

1.CA **General**.
2.CO **Normal** (**Select** panel)
3.+ Select the first point of the frame. The rectangle and the text must be entirely inside the frame.
4.+. Select the second point of the frame.
5. Right-click with the mouse
6.CO **Block**
7.> Block/Macro/Group
Select the symbol to be created.
The elements are integrated into a group.
Save the group in a new folder of the *MySymbols* symbol database**.**
8. Click the Symbols tab.
9. MySymbols
Double-click *MySymbols* to open it.
10. Right-click with the mouse.
11.CO **New Folder**
12.# Symbols for cabinets
Enter the name of the new Symbol folder.
13.+ Select all parts of the symbol using a frame.
All elements must be within the frame.
Select the first point of the frame.
14.+ Select the second point of the frame.
15.+ Drag the symbol into a Symbols library for cabinet symbols. Hold the left mouse button pressed while dragging.

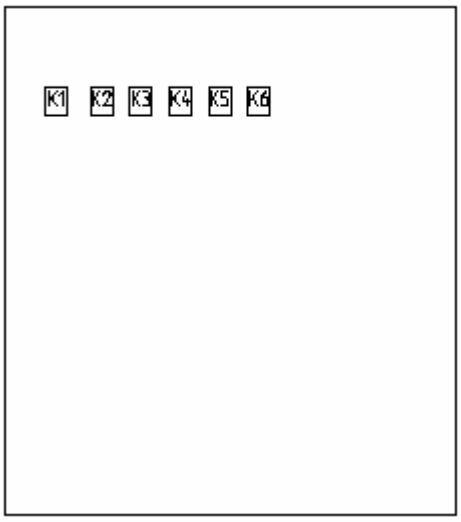
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



16.> Name
17.# Power supply 2:
Enter the name of the group.
18.> **OK**
The group is stored in the symbol database.

Exercise 40-18: Insert the component in multiple places in the panel.

For example:

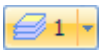


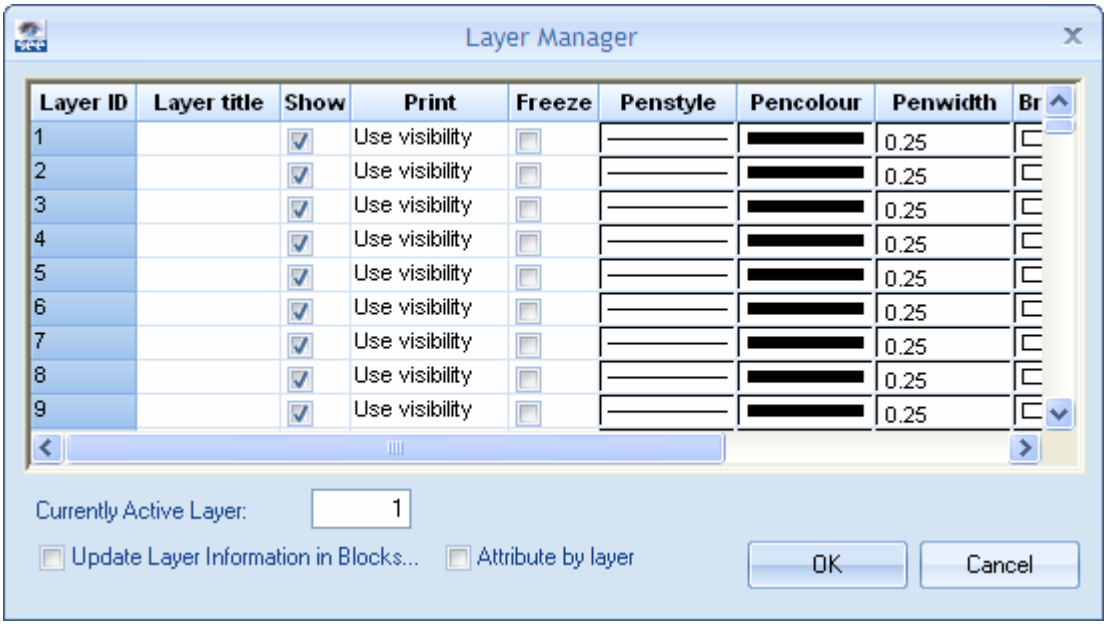
1.+ Insert the component from the symbol database into several places on the page.
2. Finish the placement.
3.+ Double-click the text and change it.
4.> **OK**
Change the next text.
etc.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

RR.2.USING LAYERS

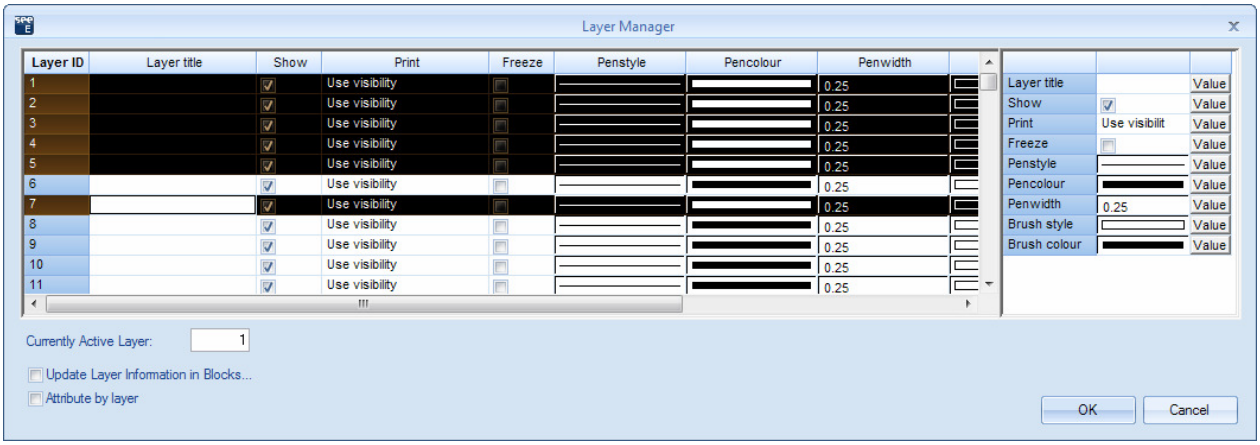
SEE Electrical provides 512 layers for structuring drawings. You can use layers, for example, for hiding some data while printing.

You can change the settings for layers by clicking on the  icon.



It is possible to select multiple layers and change their attributes in one step. The attributes can be changed for selected layers (by using the SHIFT or CTRL keys) or for all layers (by clicking on the "***Layer ID***" field).

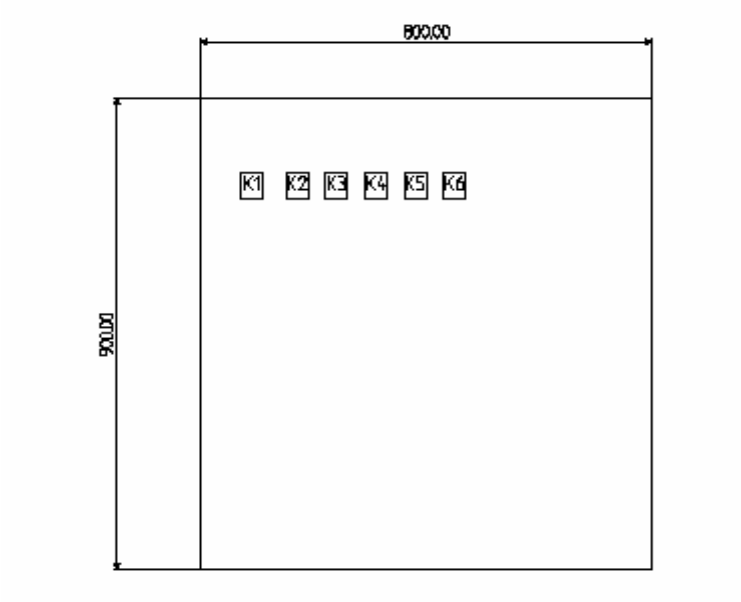
When multiple layers are selected, an editor appears in the right area of the ***Layer Manager*** window. It allows managing of the layer attributes such as visibility, color, line type etc.



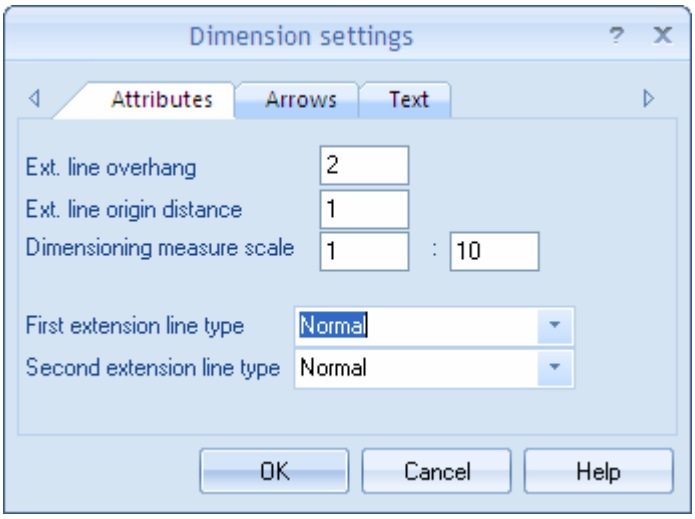
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

RR.3.DIMENSION

Exercise 40-19: Define dimensions for the panel. Adjust the settings for the dimensions.



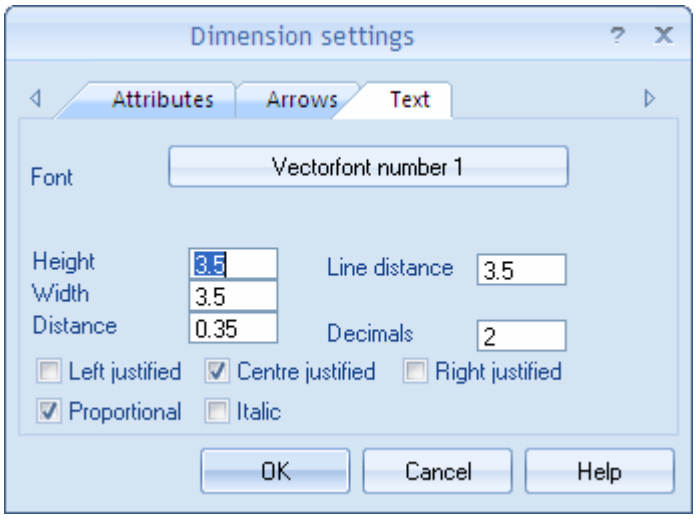
1.CA **Draw**
2.CO **Dimension**
3.CO



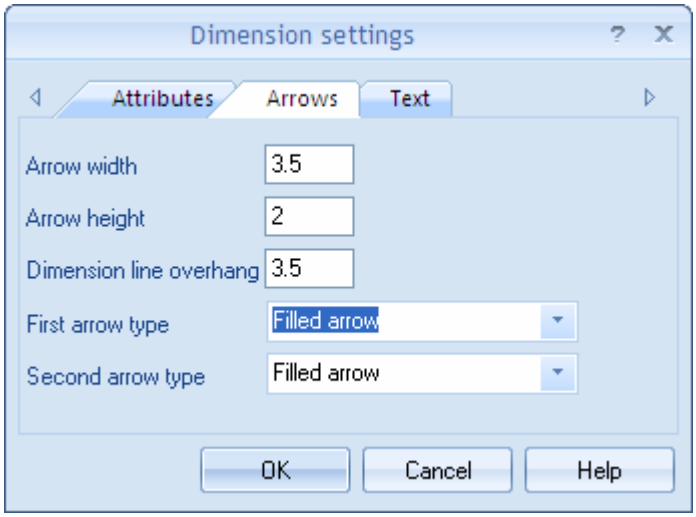
4.T Text

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Click the ***Text*** tab.



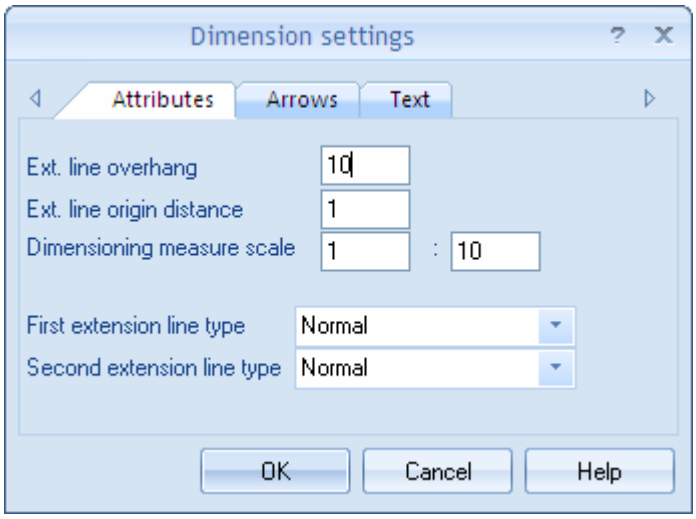
5.> Height
6.# 35
7.T Arrows
Click the ***Arrows*** tab.



8.> Arrow width
9.# 10
10.> Arrow height
11.# 10
12.> Dimension line overhang
13.# 15
14.T Attributes

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Switch to the ***Attributes*** window.



15.> Extension line overhang
16.# 10
17.> **OK**
The settings have been set.
18.CA **Draw**
19.CO **Dimension**
20.CO **Btw. 2 Lines**
21.+ Identify the left border line of the panel.
22.+ Place the dimension line by clicking with the mouse.
23.+ Identify the right border line of the panel.
24.+ Right-click to exit the drawing mode
21.+ Identify the top border line of the panel.
22.+ Place the dimension line by clicking with the mouse.
23.+ Identify the right border line of the panel.

Exercise 40-20: Save the workspace.

1.CA **File**
2.CO **Save**

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

RR.4.COMPARISON BETWEEN CIRCUIT DIAGRAMS AND CABINETS

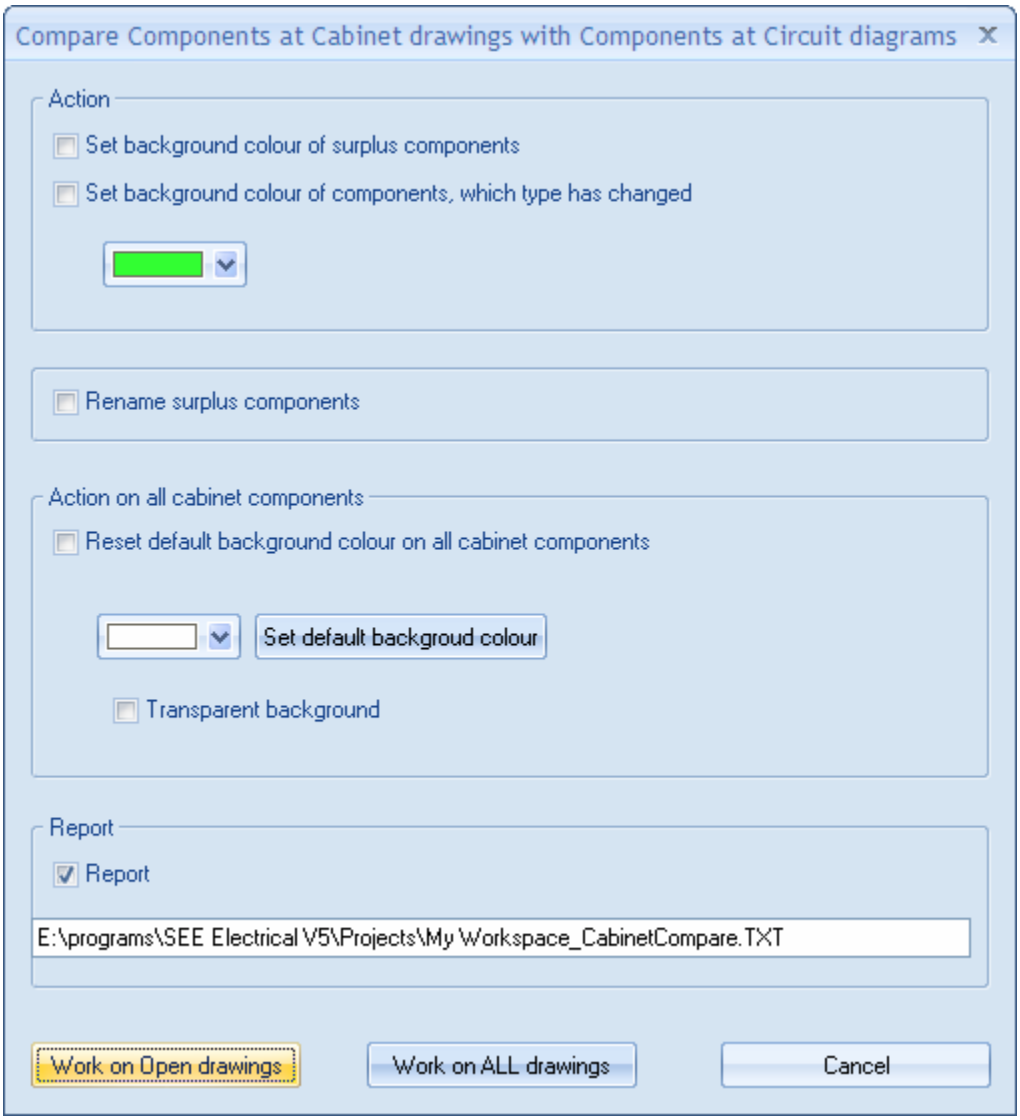
(Cabinets module)

If you have deleted components in the circuit diagram, they must also be deleted from the cabinet drawing. If types have been changed in the circuit diagram, they must respectively be changed in the symbols from the cabinet drawings.
If an existing cabinet drawing is added from one project to another project, the equipments which exist in the circuit diagram must be assigned to the equipments in the cabinet.

Exercise 40-21: Delete a piece of equipment from the circuit diagram, which you have already inserted in the cabinet drawing.

Open the page in the Cabinets module, where the deleted equipment has been inserted.

1.CA **Cabinet**
2.CO **Components Comparison** (**Functions** panel)


Compare Components at Cabinet drawings with Components at Circuit diagrams
Action
Set background colour of surplus components
Set background colour of components, which type has changed
Rename surplus components
Action on all cabinet components
Reset default background colour on all cabinet components
Set default backgroud colour
Transparent background
Report
Report
E:\programs\SEE Electrical V5\Projects\My Workspace_CabinetCompare.TXT
Work on Open drawings
Work on ALL drawings
Cancel

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

3.> Set background colour of surplus components
4.> Work on Open drawings
Click the **Work on Open drawings** or the **Work on All drawings** button to define the pages which will be processed. The differences are displayed and the corresponding components appear selected.
5.# <Del>
If you wish to delete all the marked components, press the **Del** key on the keyboard. Do not click on the drawing before that, because this will cancel the selection on all the surplus components.
The **Undo/Redo** command is available only in case you work on the open drawings. Please consider this before executing the **Cabinet** □ **Functions** □ **Components Comparison** command on your workspace.
If you do now wish to delete the surplus components, but to check them, you can reset their background colour. To do this, tick the "***Reset background colour on all cabinet components***" option in the ***Components Comparison*** dialogue. Click again the **Work on Open drawings** or the **Work on All drawings** buttons, as desired.

Exercise 40-22: In the circuit diagram, change the type for a component which has already been inserted in the cabinet drawing. Open the page in the Cabinets module where the component is inserted.

1.CA **Cabinet**
2.CO **Components Comparison** (**Functions** panel)
3.> Set background colour of components which type has changed
4.> Work on Open drawings
Click the **Work on Open drawings** or the **Work on All drawings** button to define the pages which will be processed. The differences are displayed and the corresponding components appear selected.
5.# <Del>
If you wish to delete all the marked components, press the **Del** key on the keyboard. Do not click on the drawing before that, because this will cancel the selection on all the surplus components.
The **Undo/Redo** command is available only in case you work on the open drawings. Please consider this before executing the **Functions** □ **Components Comparison** command on your workspace.
If you do now wish to delete the marked components, but to check them, you can reset their background colour. To do this, tick the "***Reset background colour on all cabinet components***" option in the ***Components Comparison*** dialogue. Click again the **Work on Open drawings** or the **Work on All drawings** buttons, as desired.

6.CA **Functions**
7.CO **Types**
8.CO **Clear Old Properties**
You must now delete the information about the modifications of the component types. Once a component type has been changed in the circuit diagram, the component is marked internally. If you have already checked the components with modified types, you need to delete the internal marking, using the **Clear Old Properties** command. After its execution, a report file appears in the *\Projects* directory with the name <Project name>_TYPE and the TXT extension. In our example, the created file is *My Workspace_Type.TXT*.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercise 40-23: Create another project, similar to your training workspace. Copy the cabinet drawing into this project from your training workspace.

1.CA **Cabinet**
2.CO **Components Comparison** (**Functions** panel)
3.> Rename surplus components
4.> Work on Open drawings
Click the **Work on Open drawings** or the **Work on All drawings** button to define the pages which will be processed.
Components that exist both in the *Circuit diagrams* and the *Cabinet* with the same database type, but with different names, are updated.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

SS USEFUL TOOLS

SS.1. COMPRESSING PROJECTS

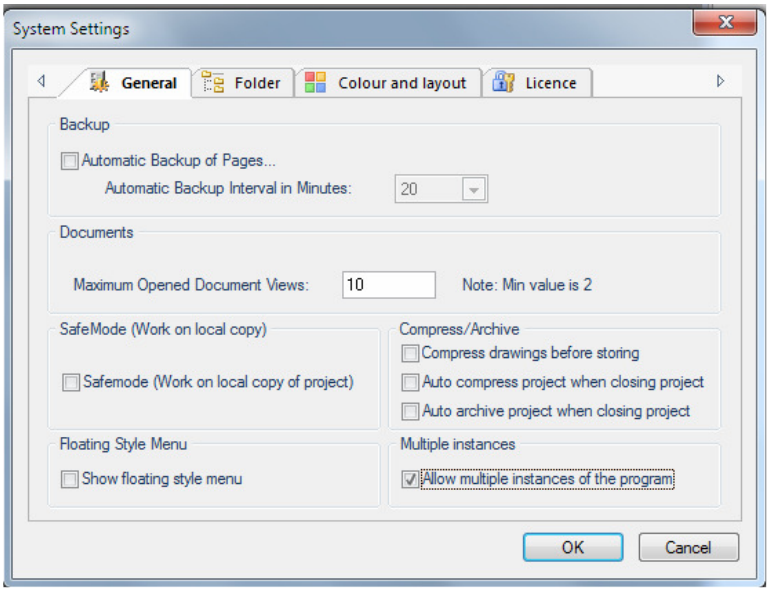
Projects in *SEE Electrical* are *MS Access* database files.

Because of the specific *MS Access* properties, the objects inserted in the database remain there even if they are deleted. They are not really removed, unless the database is compressed.
You should do this exercise periodically to avoid that your projects become extremely large and slow to manipulate.

Exercise 41-1: Compress your workspace.

1. Close all open workspaces.
   The command allowing you to compress a workspace is only accessible when all workspaces are closed.
2.CA **File**
3.Co **Compress...**
   A dialogue appears, containing all existing workspaces.
4.> <Project name>
   Select the project you wish to compress.
5.> Open
   The selected project is compressed.

It is possible now to compress workspaces each time they are closed. This functionality is activated via the "***Auto compress project when closing project***" option in the ***System Settings*** window.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

SS.2. REORGANIZE WORKSPACES

All data in *SEE Electrical* is saved as *Windows* Compound Files within a *Microsoft Access®* database. Occasionally data within the database is out of synchronization with the content of the drawings.

The "***Regenerate Database***" functionality refreshes the contents of the *Microsoft Access®* database.

1. Open a page from the workspace
2. **File Regenerate Database**
3. A dialogue box appears asking you if you want to update the database from the pages and then store your project or if you want to update the pages from the database.

Yes: Cross-references are updated automatically from the database. Entries in the database lists are created automatically, too. If you establish that errors have occurred, i.e. automatically created entries do not correspond to the information in the drawing, the database has to be regenerated. The workspace will be automatically saved afterwards.

No: If you have changed something via a database editor and your modifications do not appear automatically in the drawing, then the drawings must be updated from the database.

Cancel: Aborts the process.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

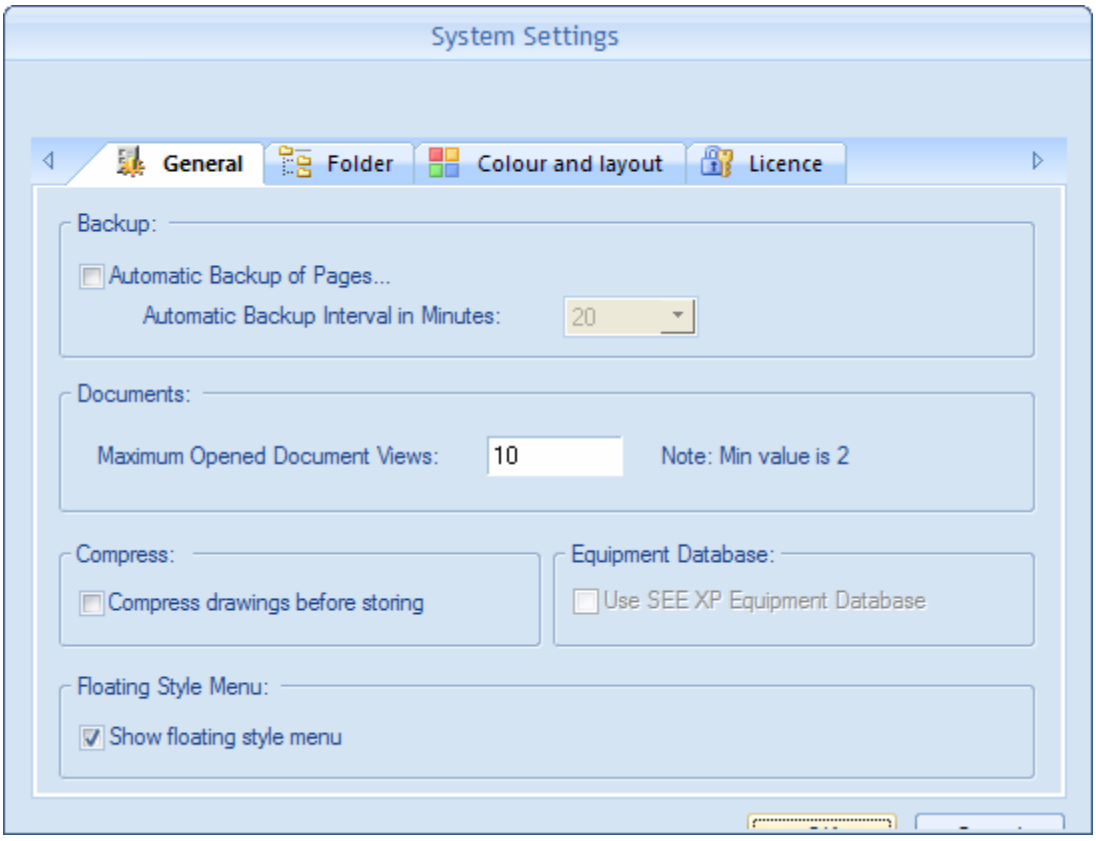
TT SETTINGS

TT.1. SYSTEM SETTINGS

The system settings apply to the system, i.e. to all projects and drawings. The system settings are stored in the *Windows* registry.

The **File** □ **System Settings** function is available only when no project is open.

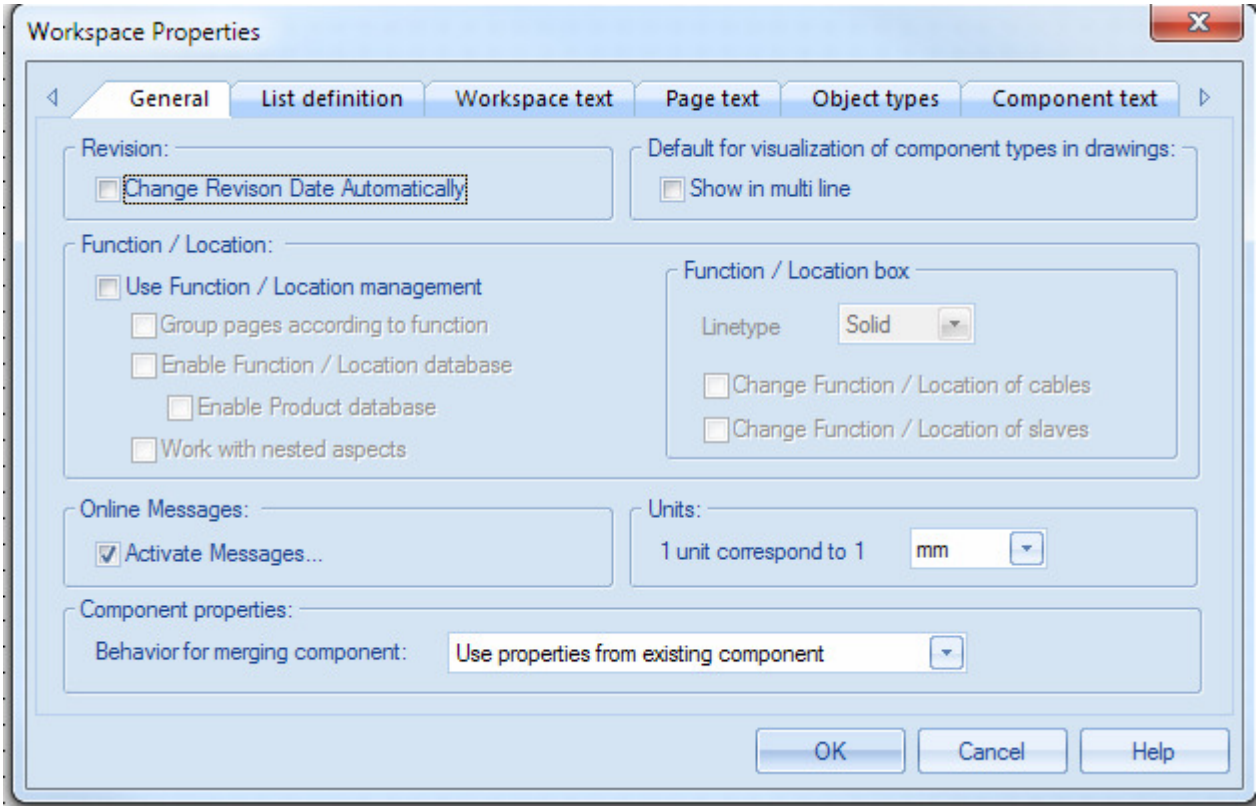
Choose, in the ***System Settings*** window, whether to make automatic Backup of Pages or not and set the maximum number of opened document views.



Click the ***Folder*** tab to define the folders where to store your templates, projects and symbols (for example, they could be stored in the network). Use the settings in the ***Colour and Layout*** tab for the colours for the background, for the cursor, etc.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

TT.2. WORKSPACE PROPERTIES



The ***Workspace Properties*** window is available for all levels.

The settings in the ***General*** tab regarding the Function/Location Management are available for the *standard* level and are described in the "*Function and Location*" chapter.

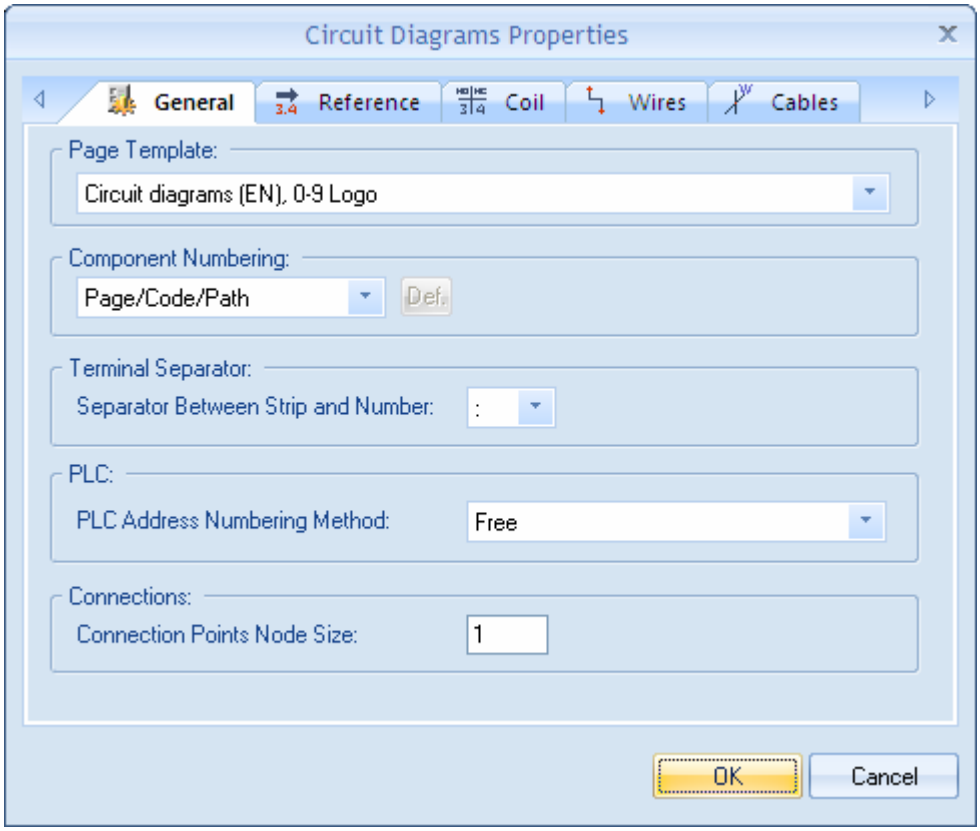
The "***Enable Function/Location database***" and "***Enable Product database***" options are available in *advanced* level and are described in the **AA. Advanced Function/Location and Product Management** chapter.

The other tabs are available for *advanced* level and are described in the "*Customizing the Workspace/Page Information Windows*" chapter.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

TT.3. PROPERTIES FOR CIRCUIT DIAGRAMS

The circuit diagram properties apply to the whole project. They are defined when you create the workspace template. The properties are loaded when you create a project by using the workspace template, and they are saved then within the project.
You can select the **Properties** function as follows:
Select *Circuit diagrams* in the ***Workspace Explorer***, then select **Home** □ **Properties** □ **Module**
or
Select *Circuit diagrams* in the ***Workspace Explorer***, then right-click with the mouse and select **Properties.**



Here you can choose the page template, the component numbering method and the text parameters for the cross-references creation.

Under *SEE Electrical standard,* you can set to display cross-references for relay coils in contact cross or contact mirror, and whether to show wire numbers in the drawing or not. You can choose which format for the cross reference will be used.

The circuit diagram properties can be changed at any time, for example, if another page template must be used for some projects. The page template will be used then for creating a new page. It is not recommended to change the setting of the component numbering during the project processing because the components numbered before making the change keep their old numbers unless they are renumbered. The modification of the *"Component numbering"* property must be made before drawing the circuit diagrams.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

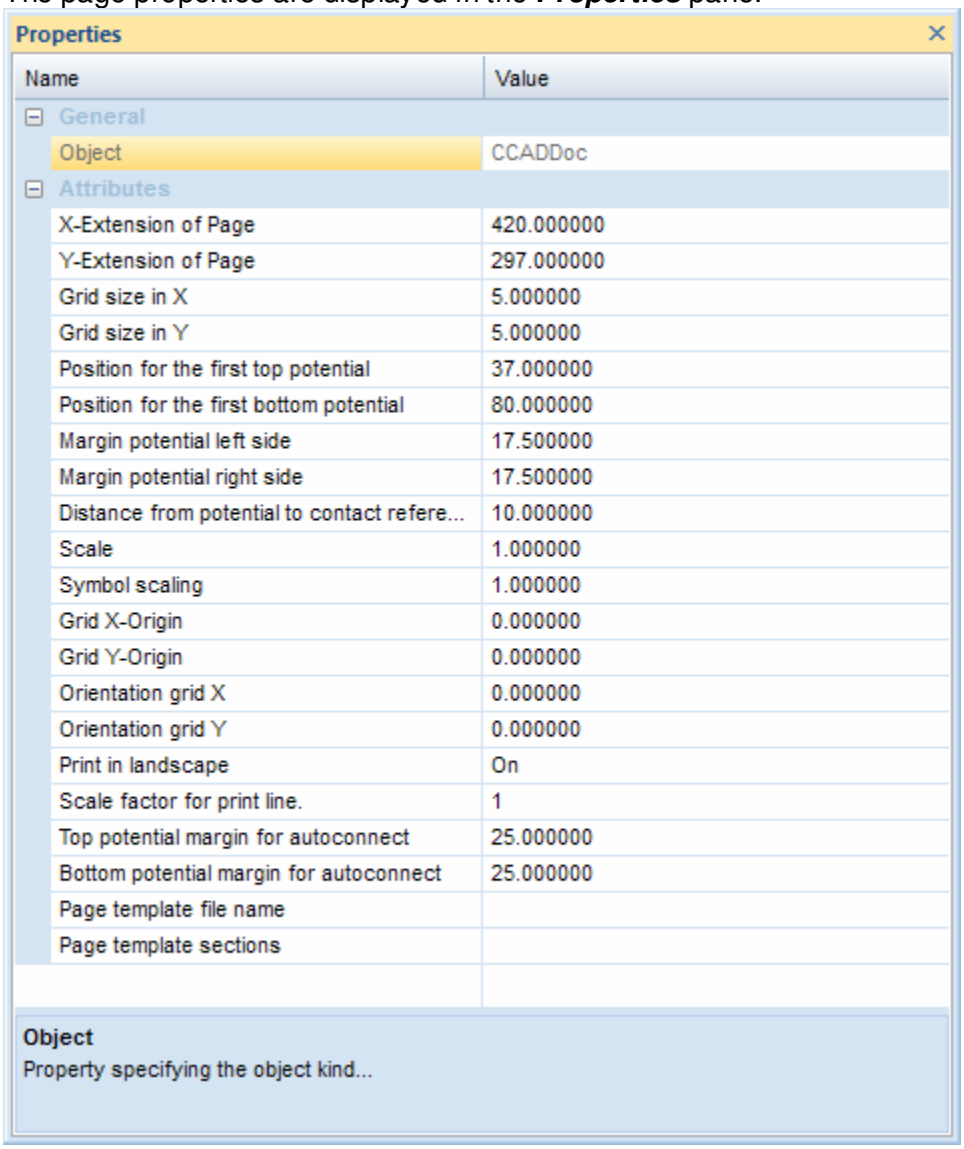
TT.4. PAGE PROPERTIES

The page properties apply to a page of the project. The page properties are specified when you create a page template. When you create a new page, the page properties are loaded from the page template and then they are stored for this page within the project.

You can select the **Page properties** function in two ways:

- Select a page under Circuit diagrams in the Workspace Explorer, and then select **Home** □ **Properties** □ **Page**.
- Select a page under Circuit diagrams in the ***Workspace Explorer***, right-click with the mouse and select **Properties…**.

The page properties are displayed in the ***Properties*** pane.



Here you can specify the number of columns in the current page and the dimensions of the drawing. It is possible to specify different numbers of columns for the different pages. The

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

dimensions of the drawing can also be different on the different pages. Of course, projects are usually constructed using the same structure.

You can set the column number in the first column (0 or 1), the margin of the left column and the margin of the right column, the positions of potentials, etc.

The grid is also defined here.
You can specify the symbol scaling, too. When you insert a symbol from the symbol database, it is scaled with the defined factor.

You can also define the page template sections.

TT.5. CUSTOMIZING THE INTERFACE

SEE Electrical allows you to customize some of the functionalities of the interface such as the application look, the commands in the ***Quick Access Toolbar***, etc.

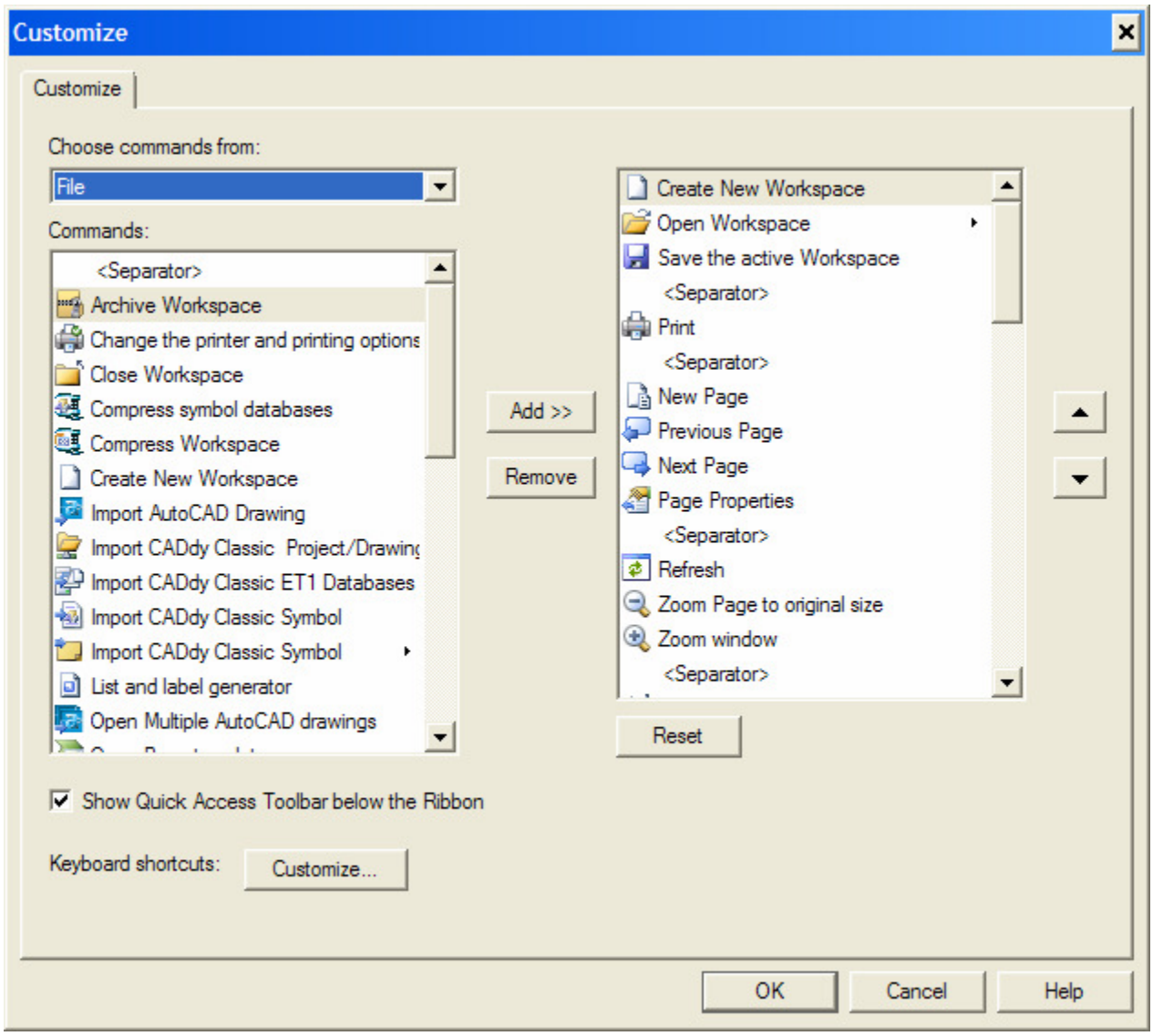
TT.5.1. QUICK ACCESS TOOLBAR

Exercise 42-1: Add the icon for the **Draw Circle** command.

1.> Right-click in the ***Categories*** area
2.CO Select the **Customize Quick Access Toolbar** pop-up command

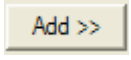
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

The following window appears:



3.> Choose the **Draw** category from the "***Choose commands from***" pull-down list.
The available commands appear in the ***Commands*** area

4.> Select the **Draw Circle** command.

6.+ Click the 

button.
The command is moved to the right area of the window.

7.> Click **OK**.
The command is now available in the Quick Access Toolbar.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

TT.5.2. DEFINE HOTKEYS

Exercise 42-2: Assign a hotkey to the command **Insert Biitmap**.

The **Customize** window that allows you to customize the Quick Access Toolbar should be open.

1.> Activate the **Customize** button.

The **Customize Keyboard** dialogue window appears

2.> In the **Categories** pane, select the menu that contains the function you want to assign a keyboard shortcut to.

3.> Select in the ***Commands*** pane the command to which you would like to assign a shortcut or whose shortcut you wish to change.

If a command already has a shortcut, it is displayed in the ***Current keys*** field.

4.> In the **Specifiy key sequence for** field select the area for which you define the key combination, e.g. for the cover sheet area or the wiring diagram etc.

5.> **New Shortcut** key

6.# <Shortcut key>

Press the desired key or shortcut on the keyboard. Letters can be combined with the keys SHIFT, CTRL or ALT.

7.> Assign

If the shortcut key is already used elsewhere, the **Assign** button remains greyed:

. In such case, you must select another key combination.

8.> Close

TT.5.3. USER DEFINED CATEGORIES

The customization is made with the *Customizer.exe* which is delivered with your *SEE Electrical* installation. The file is found in the *SEE Electrical* folder.

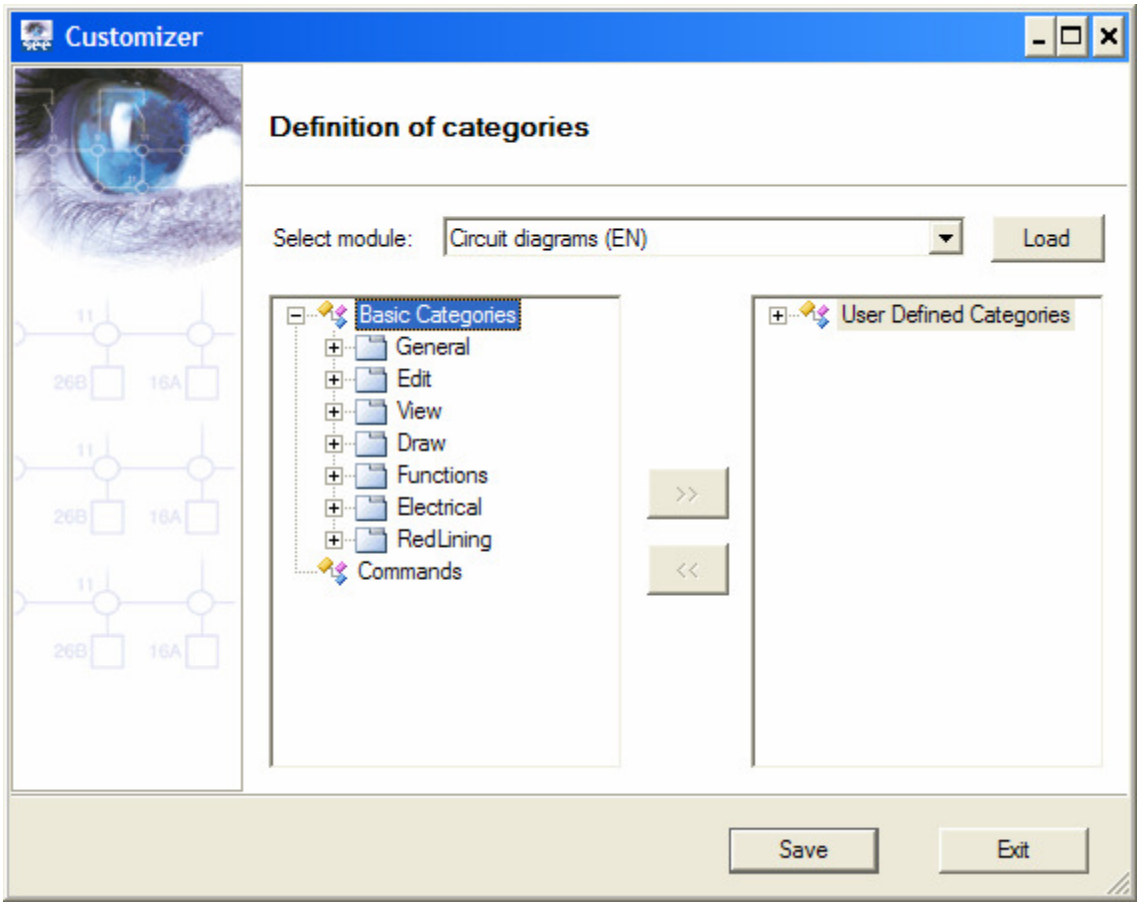
The customization can be executed in ***Basic*** level, but the commands in *SEE Electrical* will be available in case you have the right licence level.

Exercise 42-2: Create a user defined category and panel.

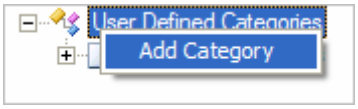
SEE Electrical must be closed before executing the procedure!

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

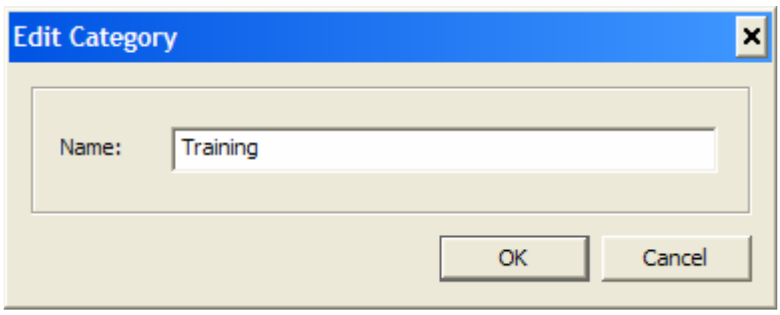
1.+ Open the *Customizer.exe*.
2.> **Next**
3.> Select the Circuit Diagrams (EN) or (IEEE) category from the pull-down list.
4.> Click the **Load** button.



5.> Right-click on the "*User Defined Categories*" node and select the **Add Category** pop-up command.,



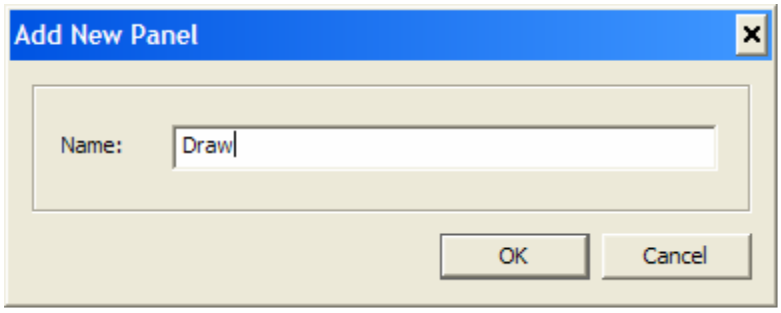
6.> Type in the name of the new category – "Training" and click **OK**.



7. To add a panel to your category, right-click it and select the **Add Panel** pop-up command.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

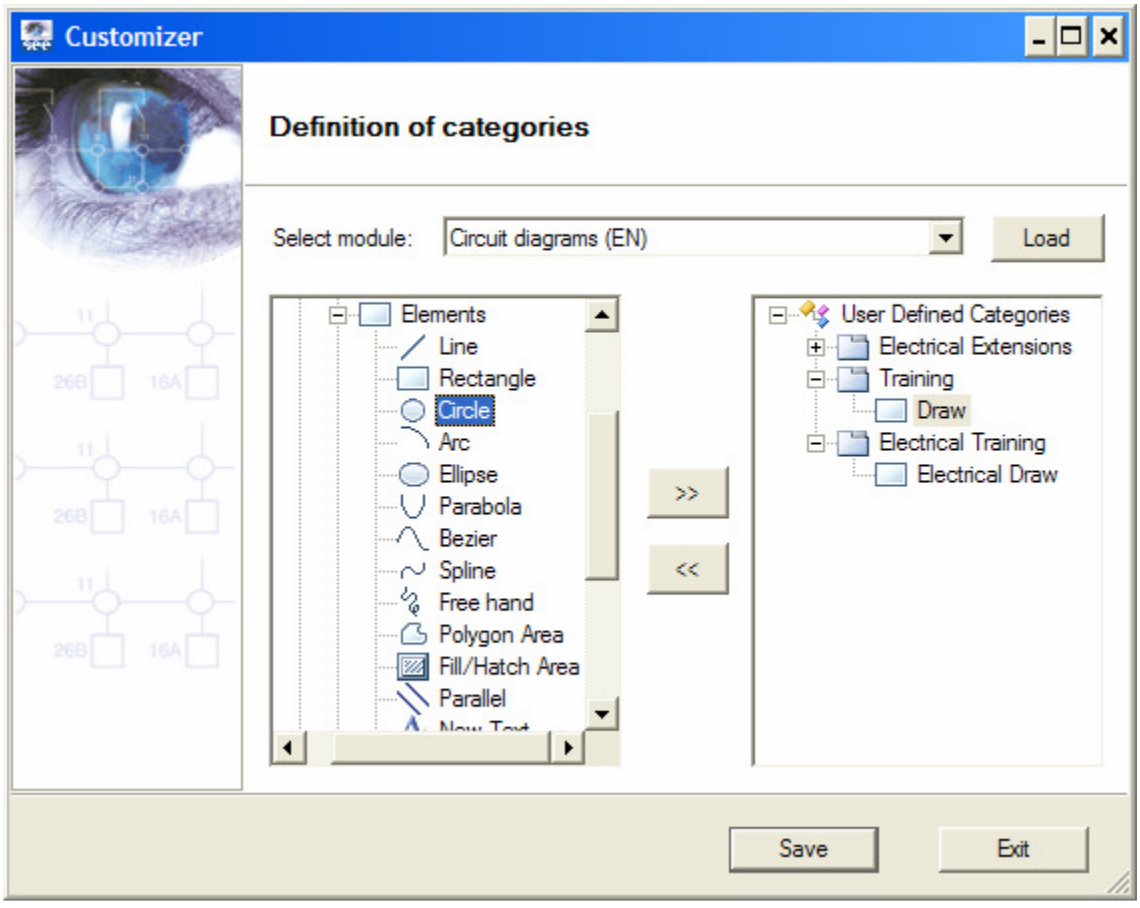
8. Type in the name of the new panel, in this case "Draw".



9. Repeat the steps to create an "Electrical Training" category and an "Electrical Draw" panel in it.

Exercise 42-4: Add commands to the user defined categories.

1.> In the ***Customizer*** window, select the **Draw** panel of your **Training** category.
2.> In the left pane of the window, explode the "*Basic Commands*" node and select the **Draw** **Elements sub-node**.
3.> Select the **Circle** command.
4.> Click the >> button.



The command is added to the user defined **Draw** panel.

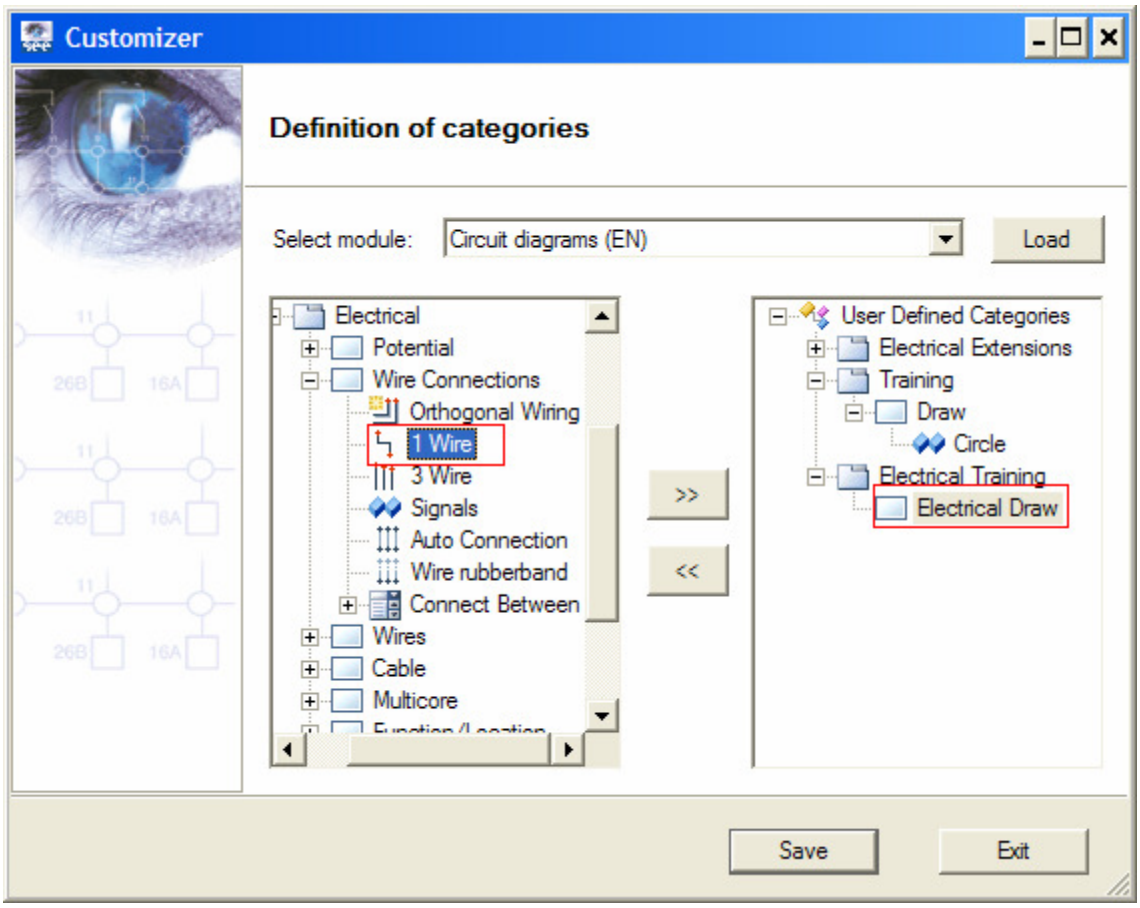
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

5.> In the ***Customizer*** window, select the **Electrical Draw** panel of your **Electrical Training** category.

6.> In the left pane of the window, explode the "*Basic Commands*" node and select the *Electrical* □ *Wire Connections* sub-node.

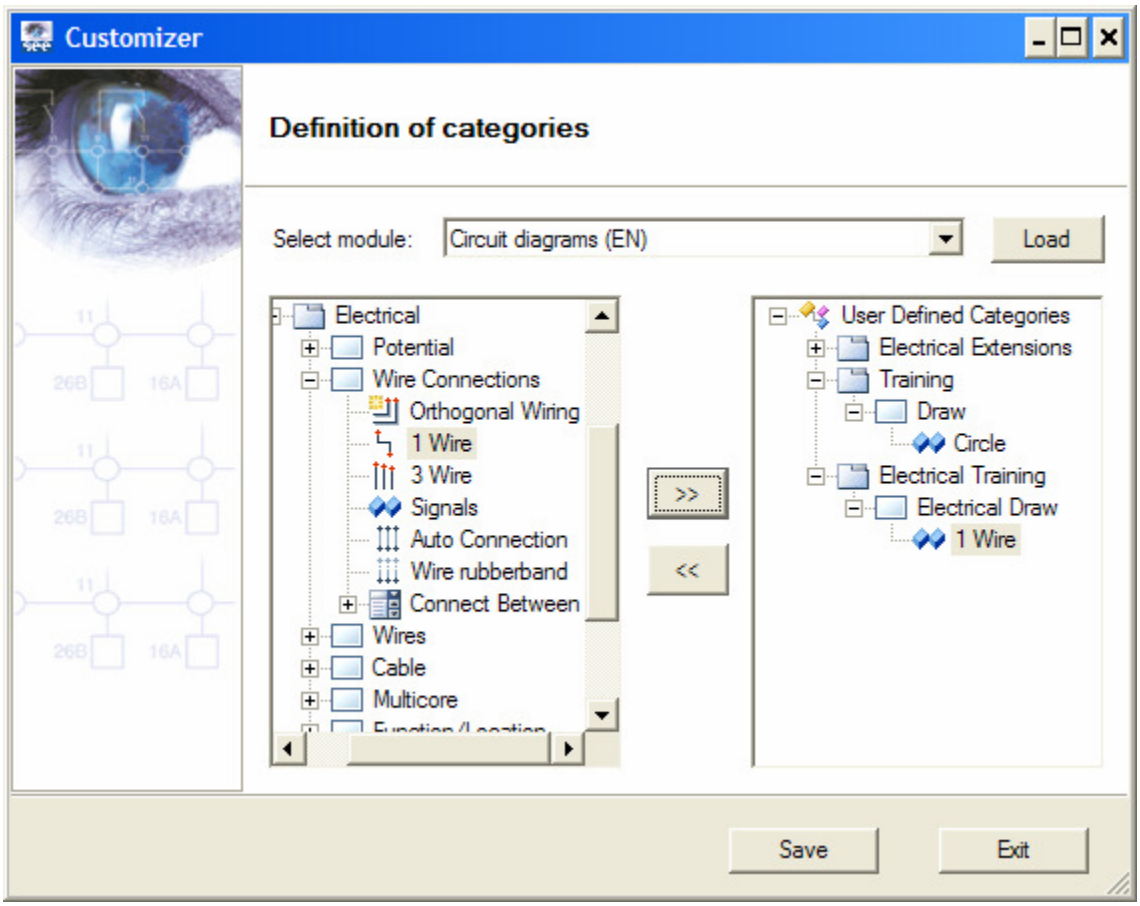
7.> Select the **1 Wire** command.

8.> Click the >> button.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

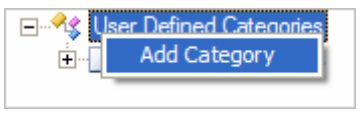
The command is added to the user defined **Electrical Draw** panel.



If you want to remove a command, select it and click the << button.

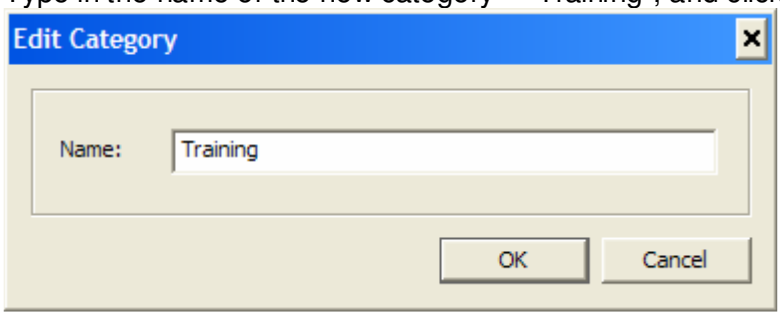
Exercise 42-5: Add commands from the ***Commands*** explorer to the user defined categories.

1.+ Open the *Customizer.exe*.
2.> Define the basic settings. All necessary files are stored by default in the *SEE Electrical* folder.
3.> **Next**
4.> Select the **General** category from the pull-down list.
5.> Click the **Load** button.
6.> Right-click on the "*User Defined Categories*" node and select the **Add Category** pop-up command.,



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

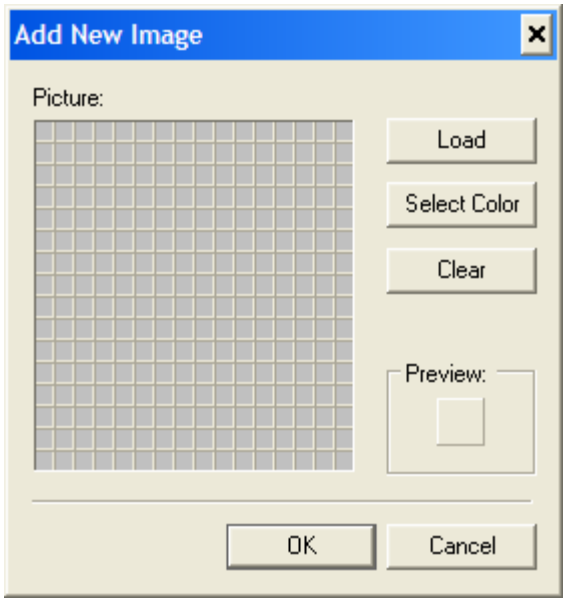
7.> Type in the name of the new category – "Training", and click **OK**.



10. To add a panel to your category, right-click it and select the **Add Panel** pop-up command.
11. Type in the name of the new panel, in this case "Tools".
13.> Explode the "*Commands*" node in the left part of the window.
14.> Scroll down to find the **ETINFO** command and select it.
15.> Select the Tools panel that you created.
16.> Click the >> button.
The **ETINFO** command is added to the user defined category.
17.> Click the **Save** button.
Open *SEE Electrical*.

Exercise 42-6: Assign/Change an icon for a command.

1.> In the ***Customizer*** window, select the **ETINFO** command in your **Training** category.
2.> Right-click and select the **Set Image** pop-up command.
3.> **New**
A new icon must be created.

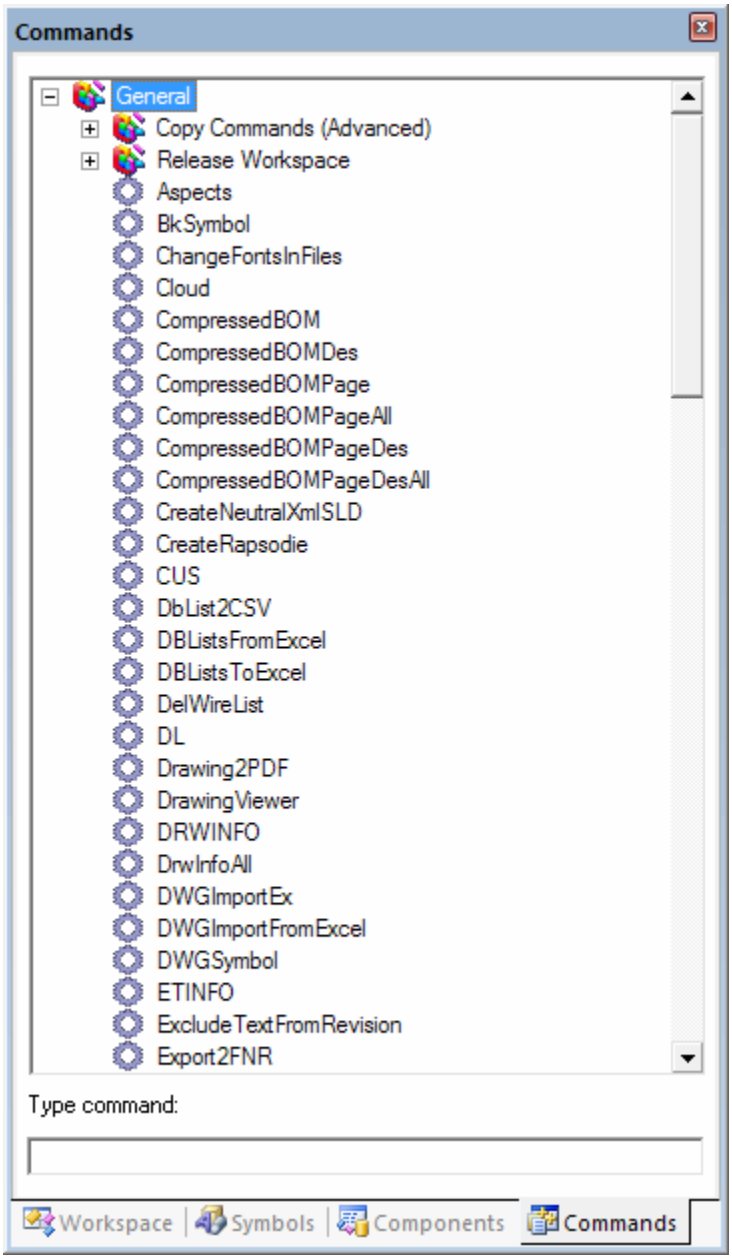


6.+ Construct a new icon. To change the default drawing colour, click the **Select colour** button.
7.> **OK**
Click **OK** to finish the construction of the new icon.
7.> **Save**.
Save the changes you made in the *Customizer*.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

UU COMMANDS EXPLORER

The ***Commands*** explorer can be used to access commands provided with *SEE Electrical* or defined by the user.
In case it is not visible, you can display it by executing the **Home** □ **View** □ **Commands** command.



The commands are displayed alphabetically. You can type in directly the name of the command in the "***Type command:***" field at the bottom of the pane and press ENTER to execute the desired function.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

It is possible also to group the commands. When you right-click on a command in the *Commands explorer* a pop-up menu appears. This pop-up menu contains functionalities to generate a group or delete one or all existing group(s). The grouping according to your needs helps to find commands quicker.

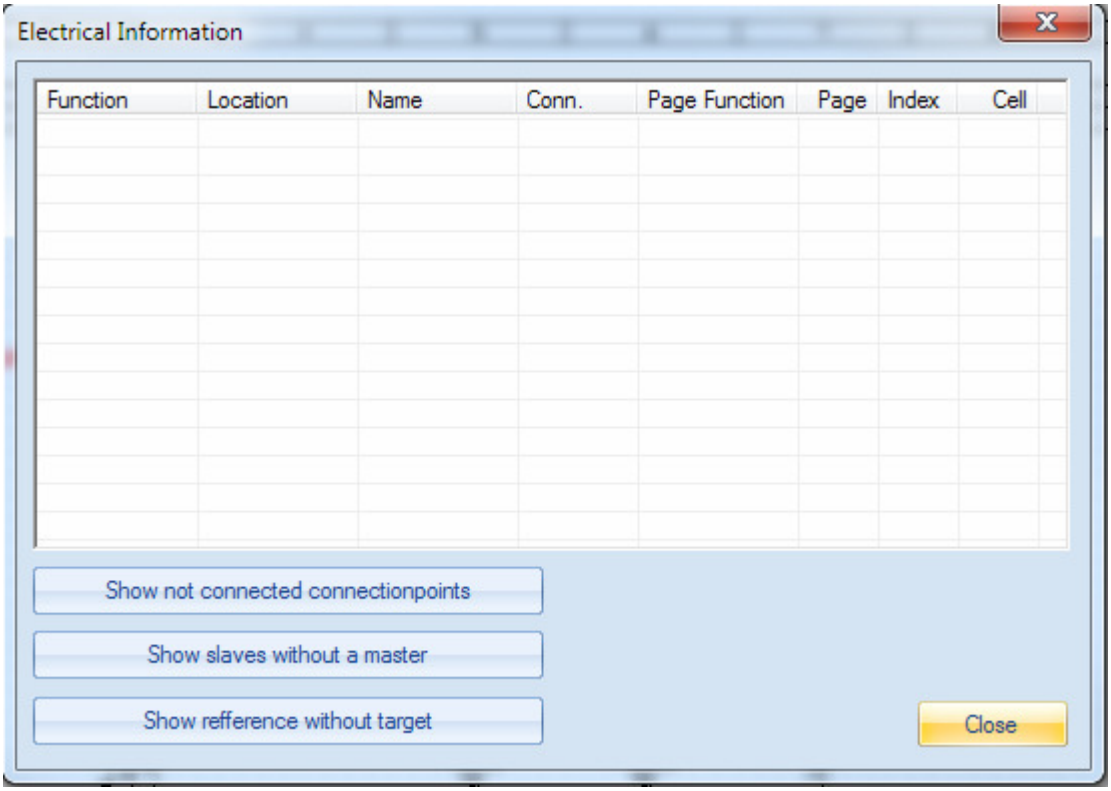
The commands can be copied and pasted to a group, they can be cut from a group and pasted to another group or they can be removed from a group, but they cannot be removed from the root level of the commands explorer. This means that you will have the commands you grouped two times: once in the root of the *Commands Explorer* and once in the selected group.

Electrical information about not connected connection points/ Information about contacts without component/ contactor

This function enables you to check if the connection points of all components in the workspace are connected.
In addition, the contacts not assigned to a contactor or to a component with auxiliary contacts can be displayed.

Exercise 43-1: Call a function from the ***Command Bar***.

1.+ Click with the cursor within the ***Command Bar***.
2.> ETINFO
Double-click the command in the list to execute it.



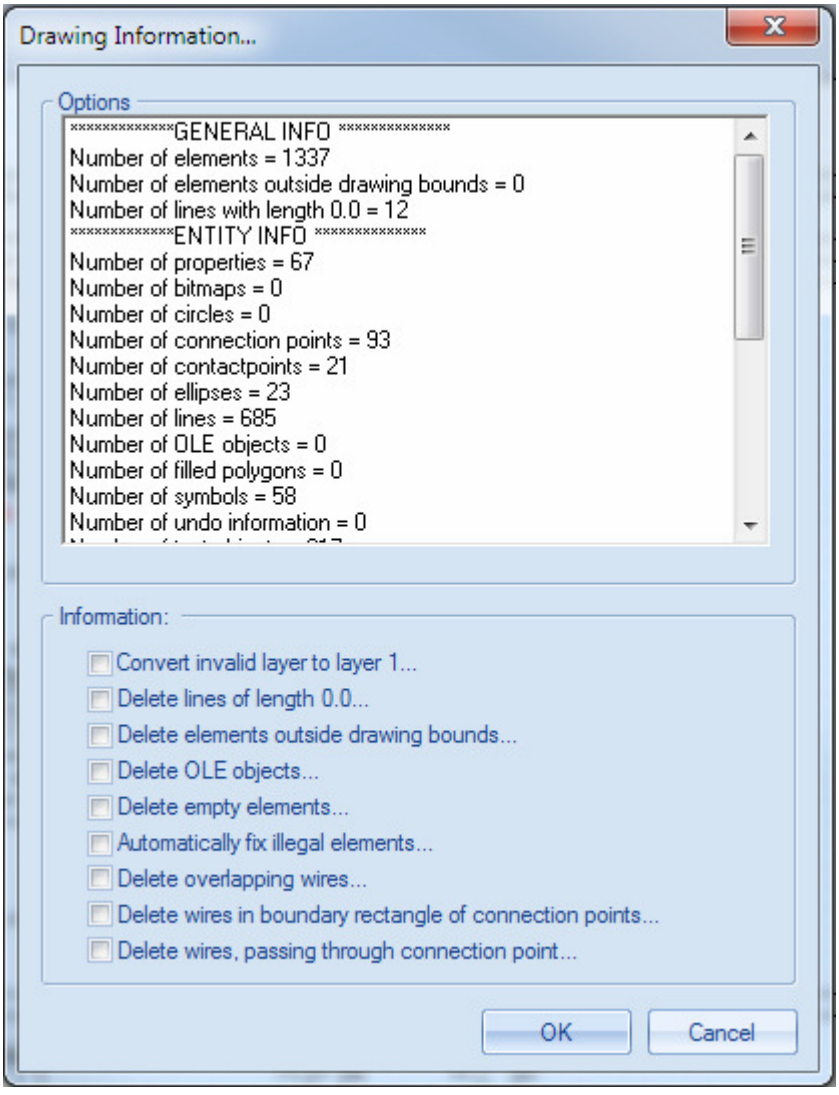
3.> Show not connected connection points
Click the button corresponding to the desired function, such as "Show not connected connection points" in the example above.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

4. Click **OK** to close the dialogue.
Delete elements outside drawing bounds
It can occur that an element is inserted outside the current drawing area (for example, by mistake while dragging). Then, you will see that nothing is shown about a component in the *Products list*.

Exercise 43-2: Call the function from the ***Command Bar***.

1.+ Click with the cursor within the ***Command Bar***.
2.# DRWINFO
Press ENTER to confirm the command


Drawing Information...
Options
**********GENERAL INFO **********
Number of elements = 1337
Number of elements outside drawing bounds = 0
Number of lines with length 0.0 = 12
**********ENTITY INFO **********
Number of properties = 67
Number of bitmaps = 0
Number of circles = 0
Number of connection points = 93
Number of contactpoints = 21
Number of ellipses = 23
Number of lines = 685
Number of OLE objects = 0
Number of filled polygons = 0
Number of symbols = 58
Number of undo information = 0
Information:
Convert invalid layer to layer 1...
Delete lines of length 0.0...
Delete elements outside drawing bounds...
Delete OLE objects...
Delete empty elements...
Automatically fix illegal elements...
Delete overlapping wires...
Delete wires in boundary rectangle of connection points...
Delete wires, passing through connection point...
OK
Cancel

3.> Delete elements outside drawing bounds.
4.> **OK**

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

Exercise 43-3: Create groups of commands in the ***Commands*** explorer

1.> Right-click on a command.

2.> Select the **New Group** pop-up command.
3.> Give a name to the group, for example, Training and click **OK**.
The group is created.
5.> Copy the desired commands by right-clicking and selecting the **Copy Command** or **Cut Command** pop-up commands.
6.> Paste the commands in the group with the help of the pop-up menu.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

VV INTELLIGENT DRAWING LEGACY MODULE

The *Intelligent Drawing Legacy* module offers you two tools that help to optimize the workflow of data imported into *SEE Electrical*.

Tools for scanned drawings:

These tools allow you to:
- ✓ import a lot of scanned drawings in one step;
- ✓ place a white background directly when a symbol is inserted.

These tools are available for all levels of the software.

Tools for making DXF/DWG imported drawings more intelligent (recognize patterns):

These tools allow you to:
- ✓ define patterns;
- ✓ recognize patterns.

These tools are available for the *Standard* and *Advanced* levels.

Tools for scanned and DXF/DWG imported drawings:

These tools are available for the *Advanced* level.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

VV.1. TOOLS FOR SCANNED DRAWINGS

VV.1.1. IMPORTING MULTIPLE SCANNED DRAWINGS IN ONE STEP

Advanced

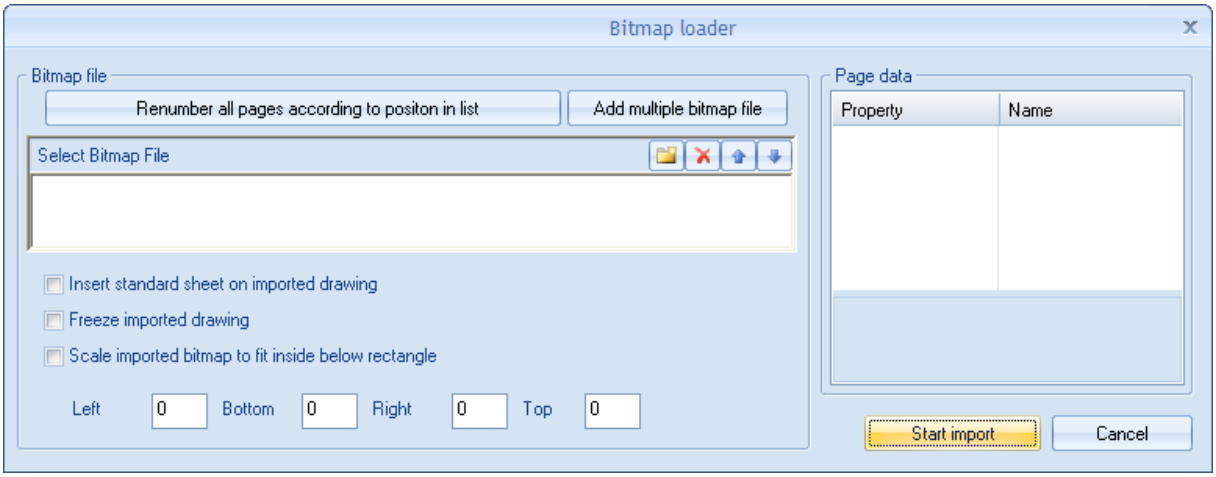
The *Scan* module allows you to import several images (of *.*TIF* and *.*JPG* format) simultaneously. The **SCANIN** command is available in the ***Command*** pane. It allows you to import different types of images and insert them as part of a workspace. With the help of this function you can insert a scanned image as a different page in the workspace.

Note:

You can use other images, often old, and insert them as part of the workspace.

Exercise 44-1: Insert scanned drawings in your project.

1. Select the **SCANIN** command from the list of commands in *SEE Electrical*.
The ***Bitmap loader*** window appears:



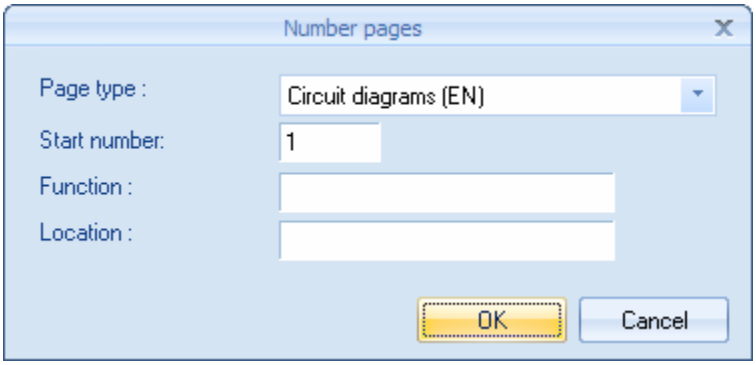
2. Click the **Add multiple bitmap file** button and select the files you want to insert in the workspace.
3. Once you have added the images, you can change the order in which they are listed via the manipulation buttons: .

Note:

The listing order is important since it allows you to define the order in which the images will be inserted in the workspace. That is to say, the first image can be inserted as page 1, the second as page 2, etc. You can also "drag and drop" the file to modify the insertion order.

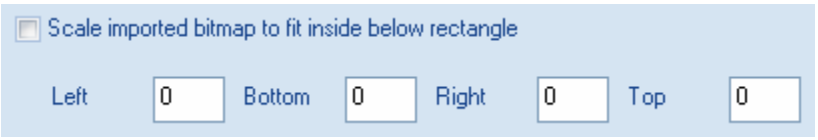
COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

4. When the order is defined, click the **Renumber all pages according to position in list** button so that *SEE Electrical* can insert a consecutive number for each scanned image.
The following dialog box appears:



5. Type in the necessary information and click **OK.**
Each scanned image can receive different settings, if you select it (it is highlighted in blue) and you modify the information in the right pane of the ***Bitmap loader*** window.
6. Tick the ***"Insert standard sheet on imported drawing"*** option to insert a default template (set as property for each type of sheet) during the image insertion.

7. Tick the ***"Scale imported bitmap to fit inside below rectangle"*** option and define the coordinates of the rectangle:



8. Click the **Import** button to start the import.

Attention:

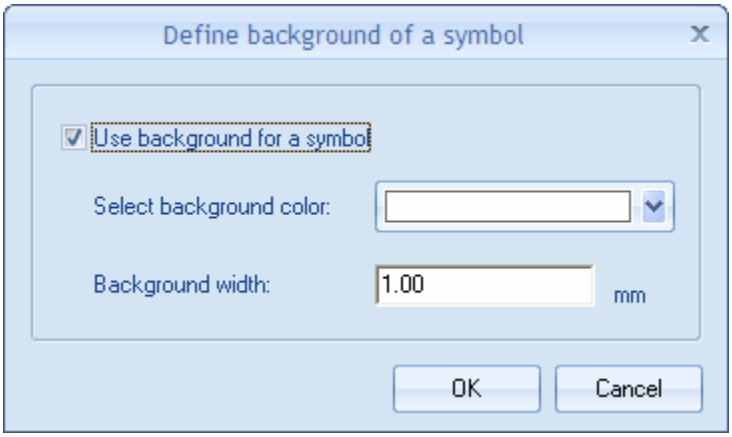
The contents of the current sheet will be deleted during the import!!! The sheet will temporarily receive the imported files.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

VV.1.2. COVERING SCANNED SYMBOLS WITH AN (WHITE) AREA ("BKSYMBOL" COMMAND)

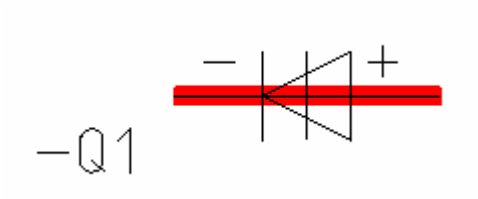
The **BkSymbol** command allows you to define if a background is to be inserted together with a symbol. This is really helpful when scanned drawings need to be processed.

1. Execute the command and define the appropriate settings.



The *"**Use background for a symbol**"* checkbox allows you to enable the functionality when you process drawings with scanned information and disable it when you process drawings created directly in *SEE Electrical*.

By default the background colour is set to white, but you can define a different colour within the *"**Select background colour**"* pull-down list.
The *"**Background width**"* option defines the width of the filling area for the background.



2. Click **OK** to validate the selected settings.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

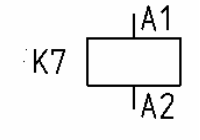
VV.2. TOOLS FOR MAKING DXF/DWG IMPORTED DRAWINGS MORE INTELLIGENT

VV.2.1. DEFINING PATTERNS ("DEFINEPATTERNS" COMMAND)

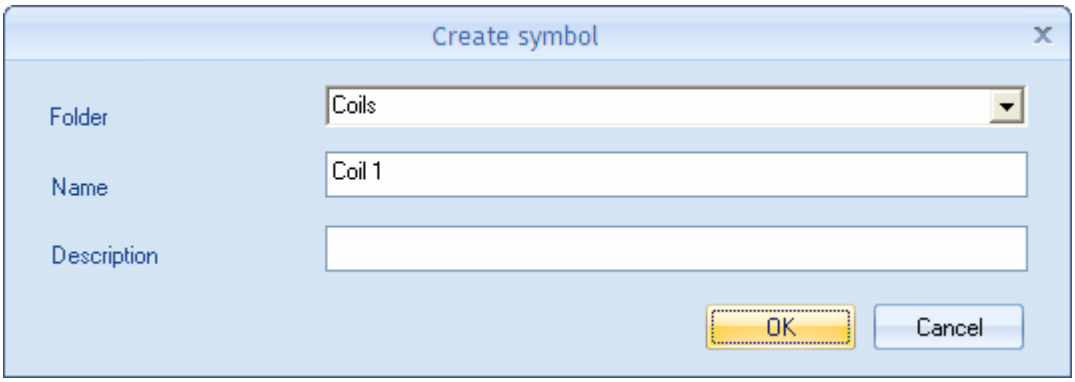
The command allows you to exchange patterns found in the diagram with symbols that are made following the rules for components, terminals, etc. Potentials and standard sheets can also be recognized.
In this way, for example, drawings imported by *DXF/DWG* import can be converted to good working *SEE Electrical* circuit diagrams that produce terminal plans, etc. This means that drawings which do not contain any logic can be changed into ones with electrical logic.
You need the specific *PATTERNS.SES* symbol library. Please get in contact, if it is missing.
The symbols can be added to existing folders only, so if you need new ones, create them before you try to define a pattern.

Exercise 44-2: Define patterns in the project.

1. Draw the geometry for the first pattern.

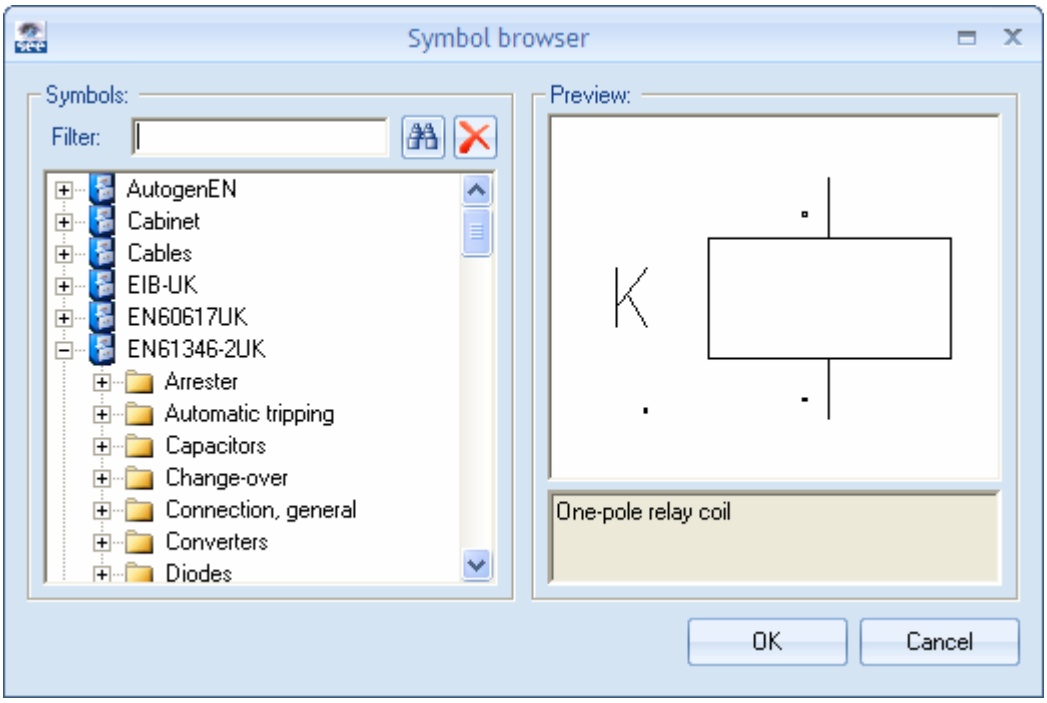


   The geometry and texts have to be single elements, not symbols or macro/groups.
2. Select the geometry/texts that shall be added to the pattern.
3. Execute the **DefinePatterns** command.
4. Define the insertion point (this will be used later as insertion point for the symbol).
5. Specify the folder and the name under which the symbol will be stored in the *PATTERNS.SES* library and click **OK**.



COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

6. Assign the *SEE Electrical* symbol that shall replace the pattern later.



7. Select the desired symbol and click **OK** to confirm.
8. Define all the patterns used in this way, including potentials.

To recognize potentials, *SEE Electrical* symbols have to be prepared inside a symbol library. This means you have to draw a potential and drag it into a folder of a symbol library. Make sure you use the same insertion point which you define later for the pattern.

Potential lines

You can also store potential lines as patterns. Before you do so, store a *SEE Electrical* potential as a symbol in one of your symbol libraries.

Standard sheet

If you want a standard sheet to be recognized, you need to store it as pattern, too. Before you do so, store a *SEE Electrical* standard sheet as a symbol in one of your symbol libraries.
Normally a standard sheet cannot be stored in a symbol library. To store it, proceed as follows:

- Take the "symbol origin" symbol from the "symbol origin" folder of your "System" symbol library. Place the symbol at the left lower corner of the standard sheet.
- Select the standard sheet and the "symbol origin" symbol.
- Click on to the "symbol origin" symbol and drag both to the folder of the symbol library where you want to store the standard sheet.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

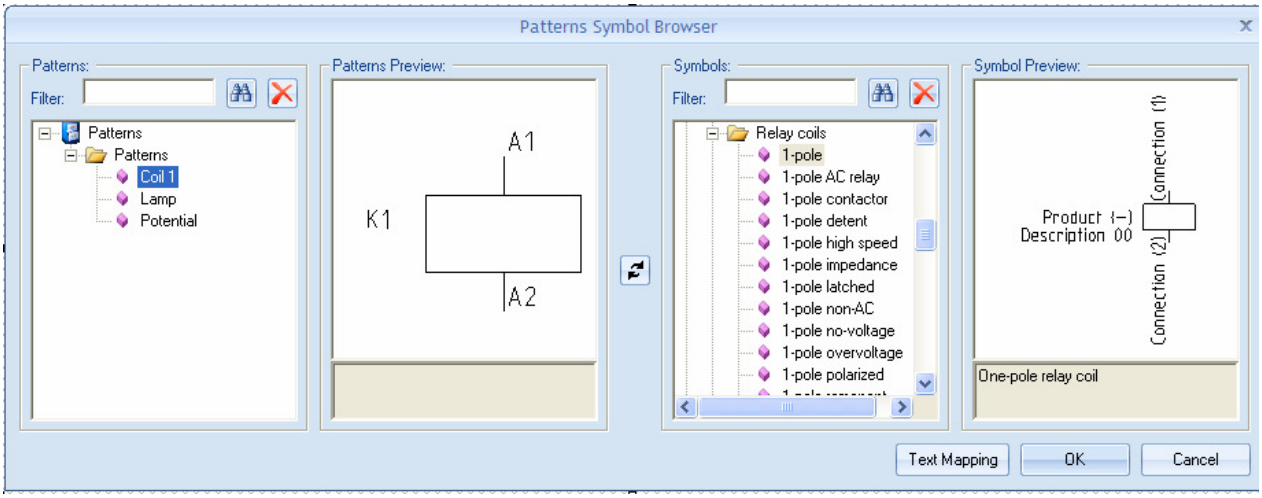
VV.2.2. CONTROLING AND CHANGING PATTERNS, ASSIGNING TEXTS ("SHOWPATTERNS" COMMAND)

The **ShowPatterns** command allows you to control and change the assignment of patterns to *SEE Electrical* symbols. It also makes it possible for the user to specify which text shall contain which information.

Exercise 44-3: Control the assignment of patterns to *SEE Electrical* symbols.

1. Execute the command.

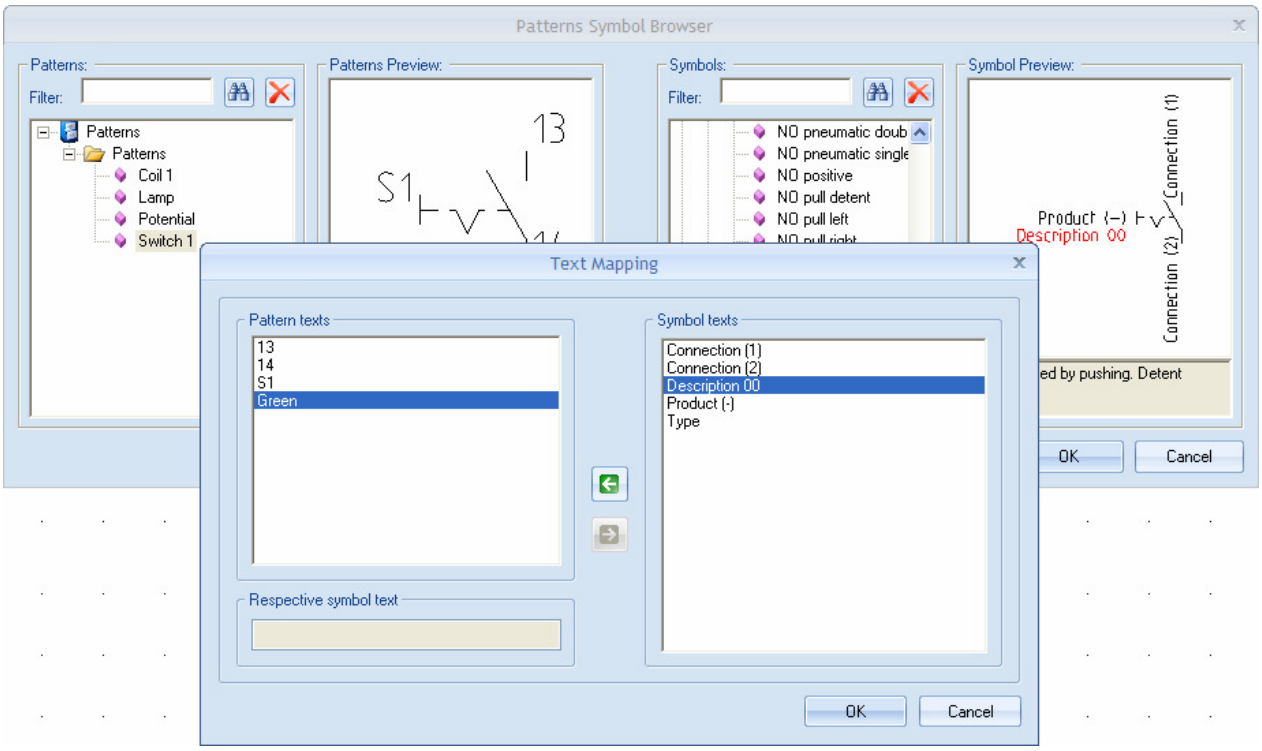
The following window appears:



The button exchanges a symbol you have chosen in the *SEE Electrical* symbol libraries with the one assigned to the pattern (the one you see in ***Pattern Preview*** pane).

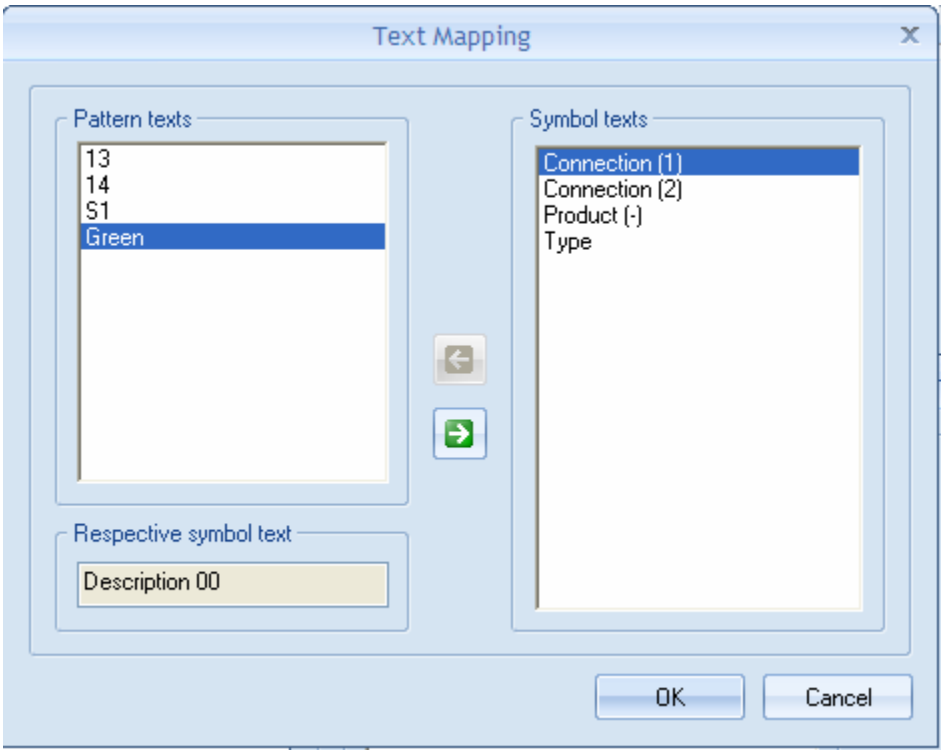
Via the **Text Mapping** button you can assign a text from the pattern to a text in the *SEE Electrical* symbol. For instance, in the example shown below you can define that the text "Green" found in a position not commonly used by *SEE Electrical* is used later as a description. All texts found in a similar position at a pattern similar to the switch will be recognized as "Description".

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados



2. Choose a pattern text and a symbol text.

Via the button you can assign them to each other. The symbol text is taken from the list on the right side. The assigned text is displayed in the *"**Respective symbol text**"* field.



3. After you have assigned the texts, click **OK** to close the window.

COPYRIGHT © 2013 IGE+XAO. Todos os direitos reservados

VV.2.3. RECOGNIZING PATTERNS ("RECOGNIZEDRAWINGPATTERNS" AND "RECOGNIZEWORKSPACEPATTERNS" COMMANDS)

Exercise 44-4: Replace the patterns in the drawing and/or the workspace with the previously defined *SEE Electrical* symbols.

1. Load the page with the drawing in which you wish to recognize the patterns.
2. Execute the **RecognizeDrawingPatterns** command.

All the patterns found in the *PATTERNS.SES* symbol library are searched in the drawing and replaced by the symbols assigned.

3. If you want to recognize all patterns found in your workspace, execute the **RecognizeWorkspacePatterns** command instead.

VV.2.4. DISPLAYING ALL ELEMENTS WITH LOGIC

By using the **MICAW** command, you can display all elements with electrical logic on the current sheet (symbols, potentials, wires and standard sheets).

Exercise 44-5: Display all elements with electrical logic in the current sheet.

1. Double-click the command in the ***Commands*** explorer to execute it.
   All logical elements are marked in green.
12. Click anywhere in the drawing to display the elements without the green marking.